

How to Use the Power Supply, Digital Multimeter and NI ELVIS

Introduction

This tutorial will introduce the usage of the power supply, digital Multimeter and ELVIS for ECE3043 laboratory class in order for students to get familiarized and comfortable using these devices.

Power Supply

The power supply used in the laboratory is Hewlett Packard 3630A Triple DC.¹

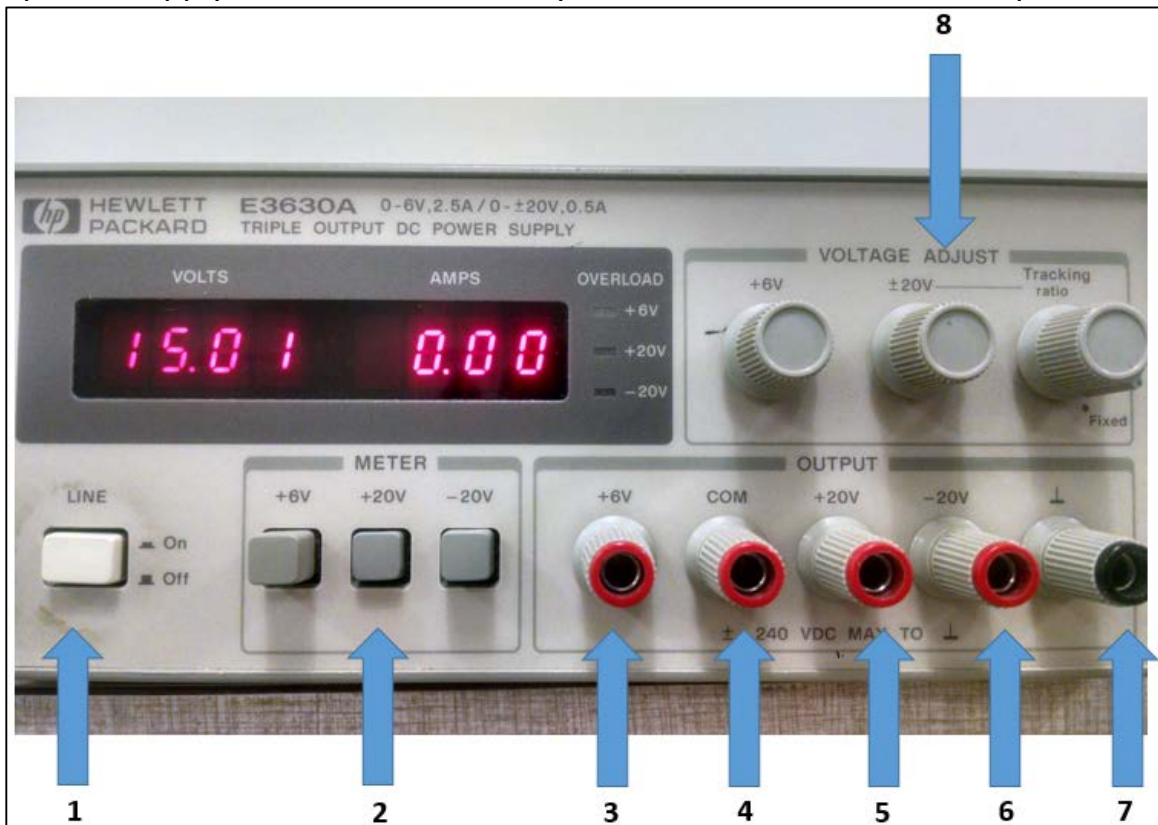


Fig. 1 – Hewlett Packard power supply with enumerated parts.

- It has three power supplies that are internally joined which produces three separate and adjustable DC voltages 6V, +20V and -20V (see Fig.1 - 3, 5 and 6).
- One output (Fig.1 – 3) varies from 0V to 6V and has a larger maximum current capacity when using with integrated circuits (ICs).
- The other two outputs (Fig.1 – 5, 6) produce \pm voltages that can be varied from 0V – 20V by using the $\pm 20V$ rotary nob (Fig.1 – 8) They are used to bias op amp circuits and/or transistor circuits.

1. The original name was Hewlett Packard which was changed to Agilent Technologies and changed again to Keysight Technologies. It is the same instrument with different names depending on when it was purchased.

- The input jack called COM (Fig.1 – 4) is shared by all three power supplies. So the + 6V output is +6V above common and the – 20V output is 20V below common, etc. In the following experiments throughout the semester, when asked to connect a pin to the ground of the power supply, it means to connect it to COM.
- The input jack on the far right (Fig.1 – 7) is normally not used and is connected to the ground wire in the AC power cord for the instrument.

Digital Multimeter (DMM)

The DMM used in the laboratory is the Agilent 34401A.

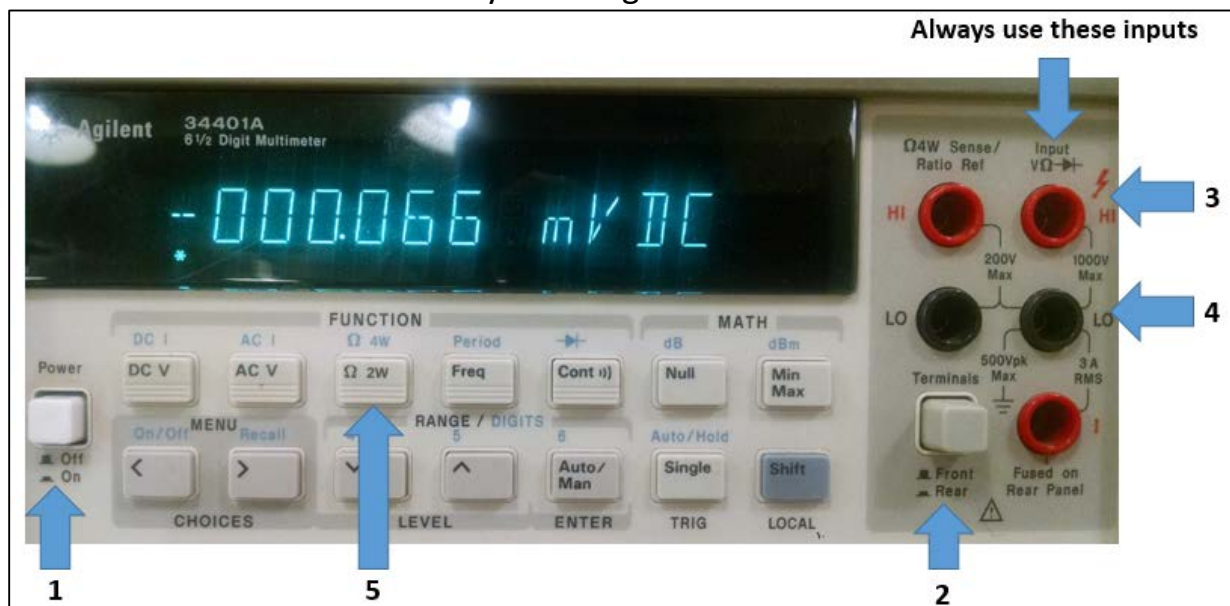


Fig. 2 – Agilent digital multimeter with enumerated parts.

- It can be used to measure DC or AC voltages and currents as well as frequencies and resistances.
- It is known as 6 $\frac{1}{2}$ digits which means that the resolution is limited to 6 $\frac{1}{2}$ digits. If there are 7 digits displayed the most significant digit (leftmost on the display) will be either 1 or 0 due to the way the instrument performs an AD (Analog-to-Digital) conversion.
- Both the power supply and DMM have female banana jack plugs. The types of leads that are used to connect circuits to either instrument are known as banana plugs (Fig.3)

For more information on:

Agilent DMM go to <http://cp.literature.agilent.com/litweb/pdf/34401-90004.pdf>

HP Power Supply got to <http://cp.literature.agilent.com/litweb/pdf/5959-5329.pdf>



Fig 3. Alligator clip to stacking Banana plug

Procedures to get familiar with the DMM and Power Supply

1. Obtain two male-to-male banana plug leads from the wall rack in the lab room.
2. Turn on the DC power supply (Fig.1 – 1) and the DMM (Fig.2 – 1). Make sure the Terminals button on the DMM (Fig.2 – 2) is set to Front.
3. Press the button +20V on the DC power supply (Fig. 1 – 2) and adjust the $\pm 20V$ rotary knob until it states 10V on the display of the DC power supply. The voltage and current being produced by the supply are indicated on the power supply.
4. Connect a lead from the + 20V jack to the COM. Note that the voltage drops to zero and the current supplied is indicated and a yellow overload light turns on.
5. Record the value of the short circuit current.

$$I_{sc} = \underline{\hspace{4cm}}$$

6. Remove the lead.
7. Connect one lead from the COM on the DC power supply to the LO (low) input of the DMM (Fig.2 – 4).
8. Connect the other lead from the + 20V jack of the DC power supply to the HI on the DMM (Fig.4 – 3).
9. Record the DC voltage indicated by the DMM and the DC power supply. If the least significant bits are varying record a typical value.

$$V_{DMM} = \underline{\hspace{4cm}}$$

$$V_{supply} = \underline{\hspace{4cm}}$$

ELVIS II+ (ELVIS II plus)

ELVIS (Educational Laboratory Virtual Instrumentation Suite) is a DAQ (Data Acquisition) device manufactured by NI that Data acquisition (DAQ) is the process of measuring an electrical or physical phenomenon such as voltage, current, temperature, pressure, or sound with a computer. A DAQ system consists of sensors, DAQ measurement hardware, and a computer with programmable software. [1]

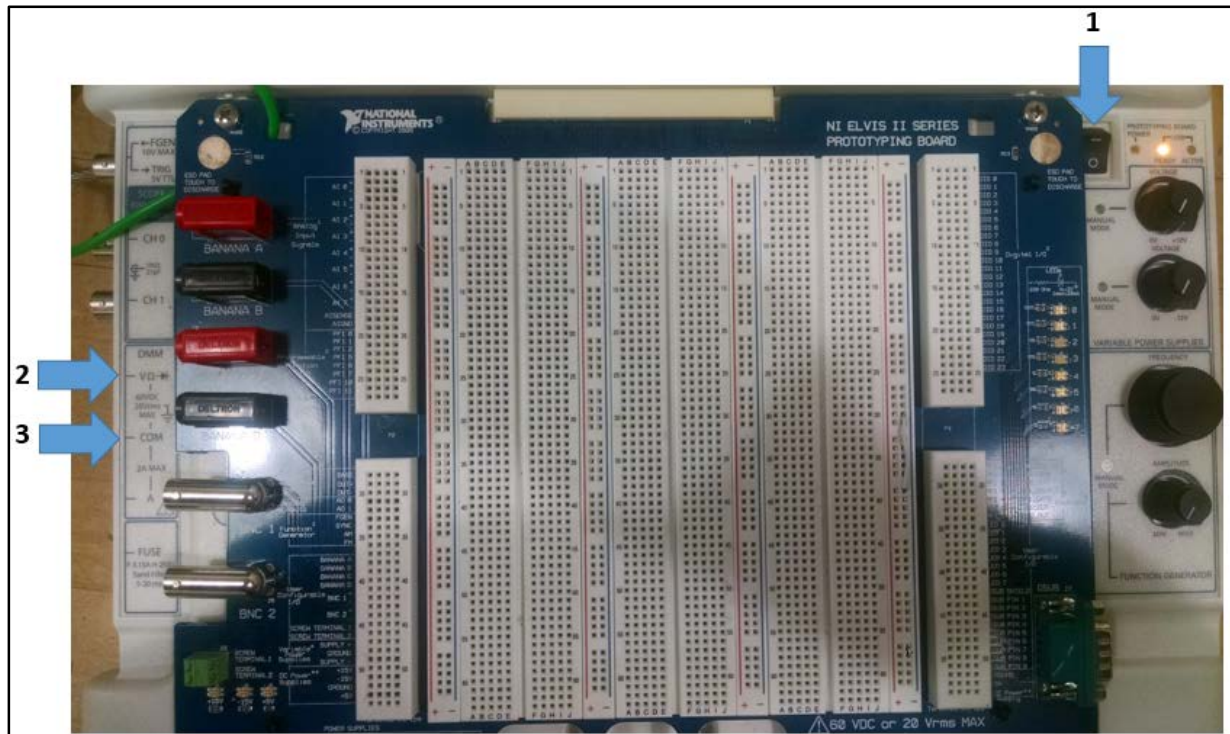


Fig 4. NI ELVIS+ with enumerated parts.

- It has ADs (Analog-to-Digital converters) and DAs (Digital-to-Analog converters) which enable it to produce and measure voltages. It can be used as a DMM, oscilloscope, RCL meter, or function generator.
- It can also be used with user written programs with their signature product the graphical program LabVIEW.
- The display for these instruments appears on the screen of a computer. This version of ELVIS is known as ELVIS II+.

Procedures to compare the NI ELVIS and DMM

1. Turn on the ELVIS workstation, there are two switches to turn **I** (**O** means off):

[1] <http://www.ni.com/data-acquisition/what-is/>

- The switch behind the ELVIS on the upper right hand (Fig.5 – 1)
- The switch on the front upper right hand corner (Fig.4 – 1)



Fig.5 Location of NI ELVIS II⁺ general turn on button.

2. Log onto the PC and start ELVIS in Start -> All Programs -> National Instruments (this is a folder) -> NI ELVISmx for NI ELVIS & NI myDaq (this is also a folder) -> NI ELVISmx Instrument Launcher -> Instruments & Apps menu -> Digital Multimeter. Note that if you do not turn on the ELVIS before you open the software, you might not get any input!
3. Connect the two leads that were connected to the Agilent DMM to the ELVIS DMM input and COM (Fig.4 – 2, 3)
4. Press Run on the ELVIS DMM front panel and record the value measured by ELVIS and the power supply.

$V_{ELVIS} =$ _____

$V_{Power-Supply} =$ _____

5. Obtain Verification Signatures from the Laboratory Instructor after the data has been recorded.

Resistance Measurement

Both the ELVIS and DMM can measure resistance. They might however get slightly different values since they evaluate resistance differently.

Procedures to compare resistance using NI ELVIS and DMM

- Remove the leads from the DC power supply and turn it off.
- Add alligator clips to the free leads and leave the other leads attached to the ELVIS.
- Connect the two leads together, press the Ohm symbol (Ω) and the null offset on the DMM ELVIS window.
- Perform the Resistance Measurement section in Experiment 1.4.2 in the lab manual.
- Measure the same resistances with the Agilent DMM by pressing the Ω 2W button (Fig.2- 5)
- Obtain Verification Signatures from the Laboratory Instructor after the data has been recorded.

How to Use the Waveform Generator and Oscilloscope

Introduction

This tutorial will introduce the usage of the waveform generator and oscilloscope for ECE3043 laboratory class in order for students to get familiarized and comfortable using these devices.

Waveform Generator (FG)

The signal source for this lab is the Agilent 33522A Function/Arbitrary Waveform Generator. It can produce basic waveforms to frequencies from a few Hertz to 30 MHz.

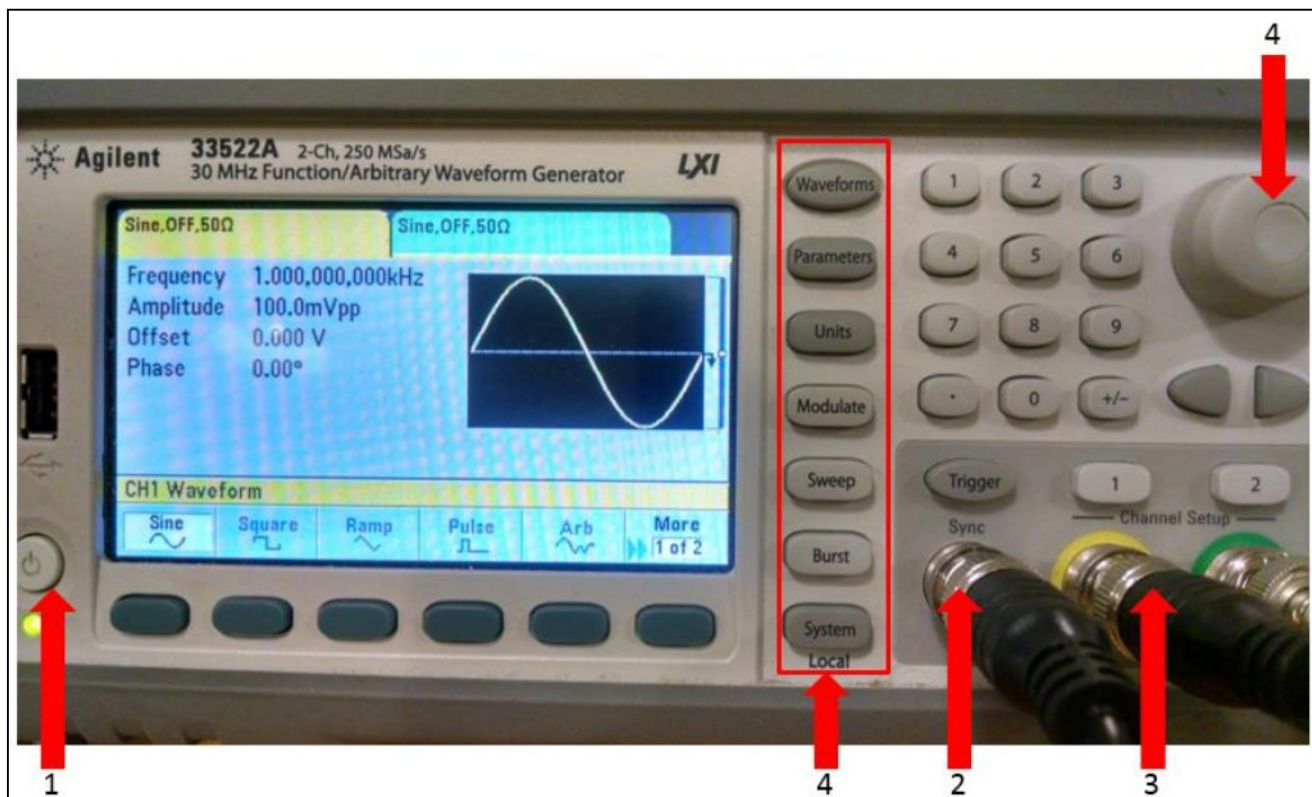


Fig 1. Agilent waveform generator with enumerated parts.

- In addition, an arbitrary waveform can be inserted into the waveform RAM memory using software.
- It also has a Sync Output, which is a TTL signal (0 – 5V) used primarily for synchronization purposes. (Fig 1-2)
- It has two separate outputs that can be independent or ganged together and the amplitude and frequency varied. The output connectors are female BNC (Bayonet N Connector) which is a cylindrical conductor system. (Fig 1-3)

Oscilloscope (Scope)

The oscilloscope for this lab is the Agilent DSO-X 3012A, a two-channel 100 MHz instrument that displays one or two signals as functions of time.

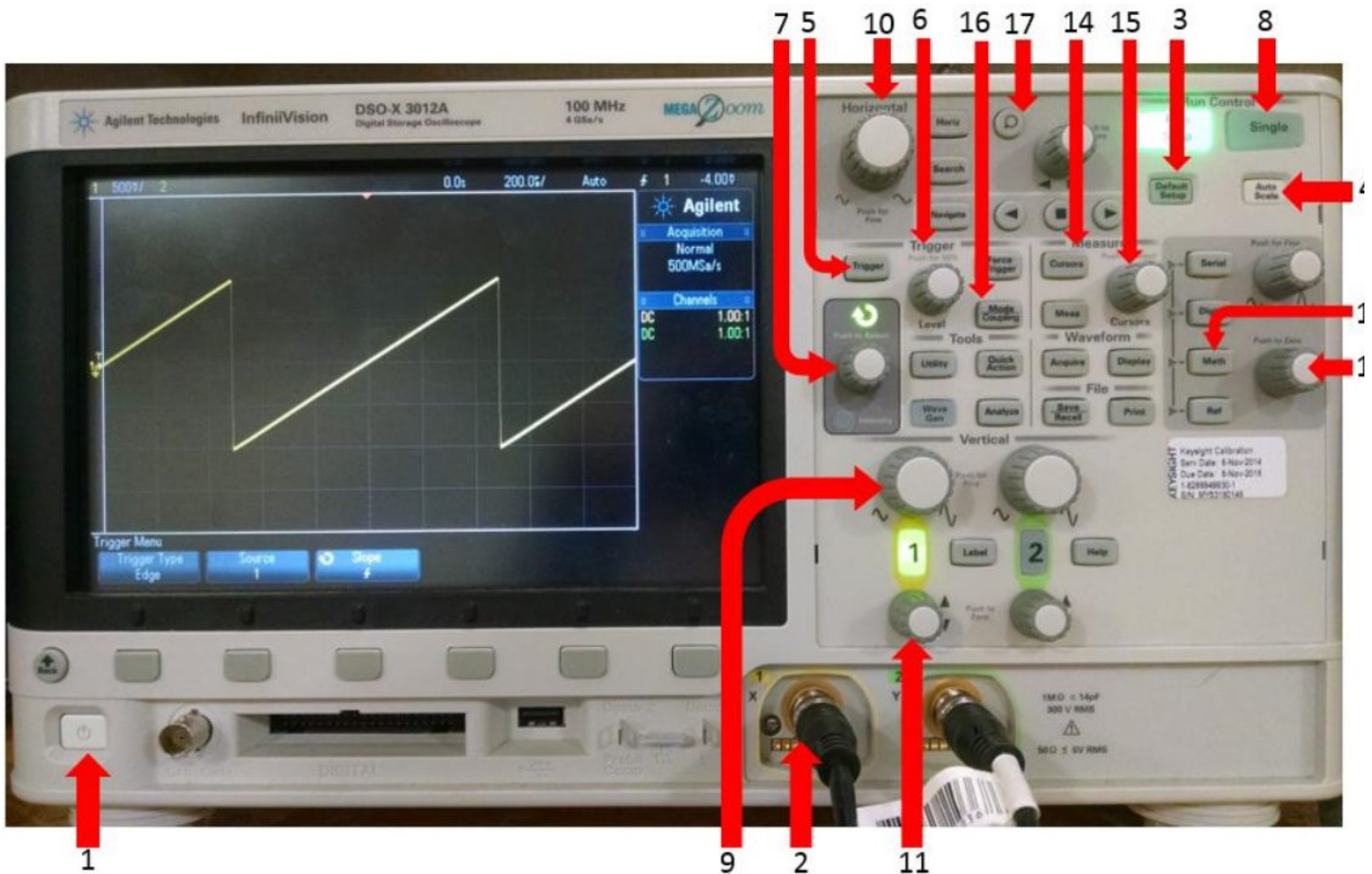


Fig 2. Agilent Oscilloscope with enumerated parts.

- It has a maximum sampling rate of 4 GSa/s. As long as the significant frequency content is below 100 MHz, the waveforms will be undistorted.
- It also has an EXT TRIG input (located on the back) that is used for synchronization purposes.
- The input connectors are female BNC connectors (Fig 2-2)

Procedures to get familiar with the Waveform Generator and Oscilloscope

1. Turn on the scope and FG with the on/off buttons (Fig 1-1, Fig 2-1). Wait for the instruments to boot.
2. Obtain a male BNC to male BNC connector from the wall rack; this is a coaxial lead with a characteristic impedance of 50Ω. (Fig 3)
3. Connect CH1 Output on the FG to CH1 input on the scope. Align the metal ridge with the metal slot and rotate clockwise until it clicks into place.
4. Do not press the button “Start 30-day Trial” on the scope and do not change the language to anything other than English.
5. Press the 1 button above the CH1 Output on the FG.



Fig 3. BNC connector

Do not connect anything to the USB connectors on the FG or scope; use the USB connectors on the PC to charge your toys.

6. On the menu that appears under the display on the FG, switch the Output to On, Output Load to High Z (High Z means the FG is to expect a load other than 50Ω which is the characteristic impedance of the coaxial lead used for high frequency applications).
7. On the buttons to the right of the display on the FG press Waveforms (Fig 1-4) and select sine.
8. Press Parameters (Fig1-4), change the frequency to 1 kHz (use the numeric pad to the right of the display), and Amplitude to 2Vpp (voltage peak to peak - the difference between the top and bottom of the waveform, this is only correct if High Z was set).
9. On the scope press the Default Setup button (Fig 2-3) and then press Auto Scale (Fig 2-4)
10. On the scope, make sure the Ch1 button is illuminated. Then press it to change the menu under the display. Which should indicate the coupling for CH1 and input impedance of $1M\Omega$ (don't change either).
11. Note that the display states in the upper left hand corner which channel is active, the VOLTS/DIV for that channel (there are 8 major vertical divisions) and the TIME/DIV (there are 10 major horizontal divisions).
12. On the left at the beginning of the sine wave there is a small T indicating the trigger setting indicating that the scope is being triggered by the waveform on CH1, and the trigger level. The absence of the T means that the scope is being triggered by the external input and not Ch1. You can change it by pressing the Trigger button (Fig 2-5) and changing the Source from Ext to 1 on the screen.
13. In addition, there is a small picture of a ground indicating where 0 volts is located.
14. To the right of the display it states that the current sampling rate (Sa/s), the trigger mode (Normal), and that each channel is DC coupled and set for a times 1X probe.
15. On the FG press Waveforms and select square.
16. On the FG press Duty Cycle, vary the rotary knob (Fig 1-4) and note and record the effect. The duty cycle is the percentage of the time the waveform is at the upper level.

-
-
17. Press Waveforms, select Ramp, and then Symmetry. Use the rotary knob to note the effect of varying the symmetry. What value of symmetry turns the ramp into a triangular wave?
-
-

Trigger Control

18. The Trigger controls determine where $t=0$ is on the display of voltage versus time. It begins in the middle of the screen with the voltage level and the slope selected.
 19. Press symmetry and select 100 % for the ramp output of the FG.
 20. On the scope vary the Trigger Level knob (Fig 2-6) and describe what happens to the display and what happens when the trigger level is set above the positive peak of the waveform.
-
-

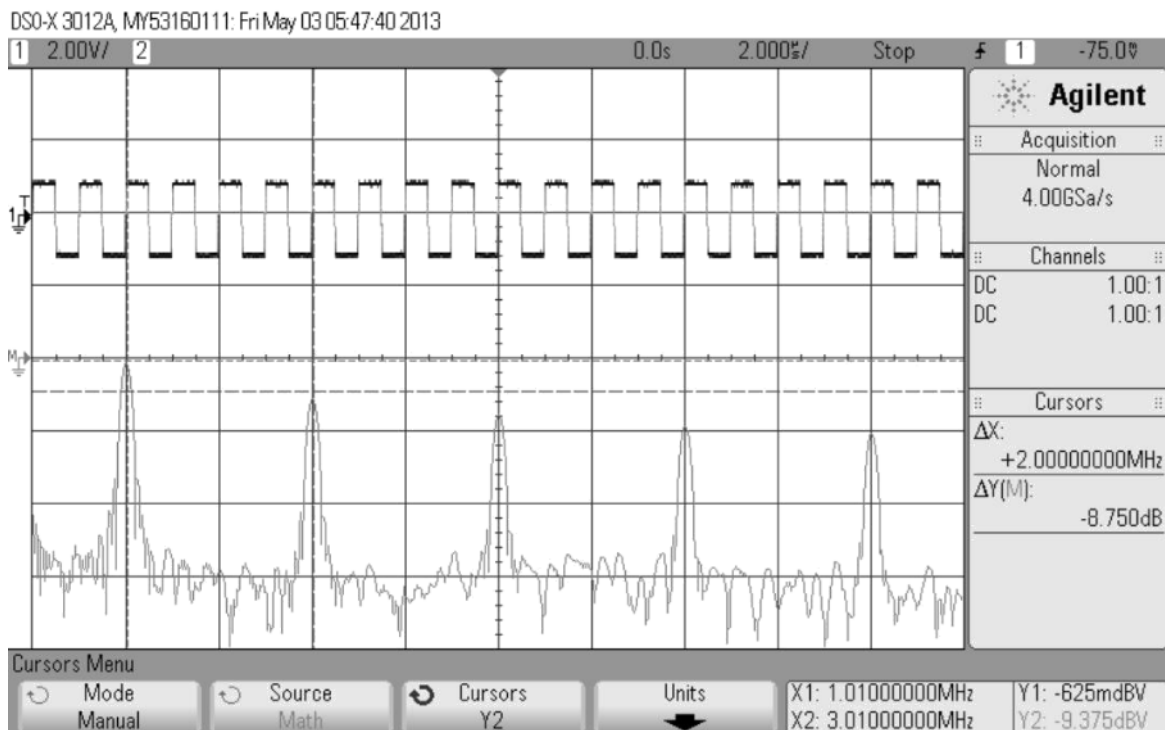
21. Press the Trigger Level knob to set it to 50% for the trigger level.
 22. Press the Trigger button (Fig 2-5) and then the Slope button on the soft menu under the display.
 23. Select the Falling slope by either pressing the slope button again or by rotating the knob with the green circular arrow on top of it (Fig 2-7) and then press it to select. Note the effect on the display.
-
-

24. Set the trigger slope back to positive or Rising.
 25. On the FG press Waveforms, More, PRBS (Pseudo Random Binary Sequence).
 26. Press PRBS Data on the FG and select PN7.
 27. Press Auto Scale on the scope and then SINGLE (Fig 2-8)
 28. Press SINGLE several times and note what happens. Write the Bit Rate indicated on the screen of the FG.
-
-

29. Print this display by downloading and running the Oscilloscope Screen Capture Program on the class web page under software applications.
30. Click on the Capture Screen Image button, put the pointer in the middle of the screen shot, right click it, and select copy data.
31. Download the Laboratory Report Template from the class web page. Paste the screenshot into the open space to the right of the picture.
32. Replace Tom Brewer's picture and name in the first square with your picture (bitmap) and your name.

Spectra of Square Wave

1. Press Waveforms on the FG, select Sine wave, Parameters, and set the frequency to 1 MHz and the amplitude to 1Vrms (Voltage root mean square).
2. Press Auto Scale on the scope.
3. On the scope, manually reduce the Volts/Div to 2V with the Ch1 Level knob (Fig 2-9)
4. Manually reduce the Time/Div to $2\mu\text{s}$ per division by using the large Horizontal knob (Fig 2-10)
5. Use the Vertical Position Control knob for Ch1 (Fig 2-11) to position the waveform near the top of the display.
6. Press the Math button (Fig 2-12) and on the display change the Operator to FFT (Fast Fourier Transform).
7. Press the Push to Zero knob (Fig 2-13), then on the display change the Span to 10 MHz and the Center to is 5MHz. Both of these use the knob with the green circular arrow on top of it (Fig 2-7).
8. Print the display and put it in your lab report.
9. Change the waveform to square on the FG.
10. Press the Cursors button (Fig 2-14), change the Source on the display to Math f(t).
11. Use the Cursors knob (Fig 2-15) to position the Cursor X1 to the fundamental at 1MHz.
12. Select Cursor X2 on the display and position it at the harmonic at 3MHz.
13. Select Cursor Y1 on the display and position it at the top of the fundamental at 1MHz.
14. Select Cursor Y2 on the display and position it at the top of the harmonic at 3MHz.
15. Print the display and place it into the lab report template. It should look like.



16. Compare the theoretical and experimental spectra for the square wave.

17. Turn off the cursors by pressing the Cursors button (Fig 2-14). Turn off the FFT by pressing the Math button (Fig 2-12).

AM (Amplitude Modulation)

1. On the FG, press Waveforms and select Sine wave, press Parameters, set the frequency to 5MHz and the amplitude to 1Vrms.
 2. Press Auto Scale on the scope.
 3. On the FG, press Modulate (Fig 1-4) and turn it On.
 4. Change the Type to AM and change the AM Depth to 50%.
 5. Select More and change the AM Frequency to 10kHz.
 6. On the scope, manually change the Time/Div with the Horizontal knob (Fig 2-10) to 20 μ s/Div.
 7. Vary the Trigger Level knob (Fig 2-6) and change the trigger level (the little T that is to the left side of the display, if it is not there, repeat step 12.) so that it is about $\frac{3}{4}$ the way up the display between the peak and trough of the envelope.
 8. Press the Trigger Mode Coupling button (Fig 2-16) and on the display increase the Hold off to about 60 μ s. A stable display should result.
 9. Print the display and place in lab report.
 10. On the FG press AM Depth and observe the effect of varying it from 50 to 80% using the rotary knob (Fig 1-4). Reset the AM Depth to 50%.
-
-

11. Change the Volts/Div, Time/Div, and Vertical Position for Ch1 on the scope until about 10 cycles are displayed near the top of the display.

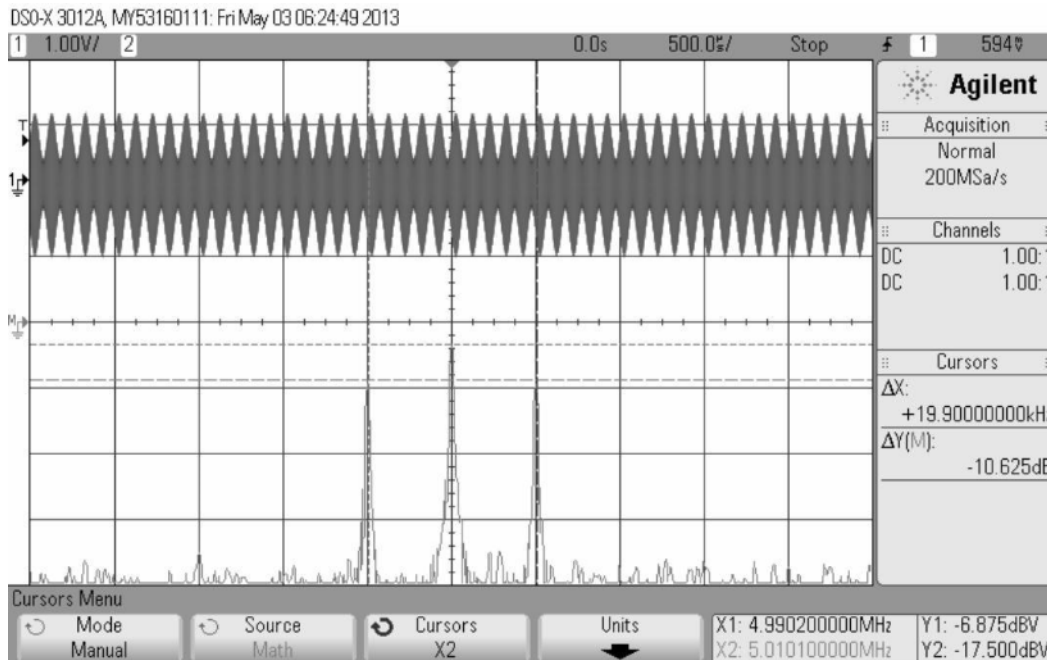
12. Press the Math button to turn on the FFT (Make sure on the display the Operator is FFT).

13. On the display, set the Center to 5MHz, and the span to 100kHz.

14. Press the Push to Zero knob (Fig 2-13).

15. Change the Time/Div with the Horizontal knob on the scope until the three spikes look reasonable like straight lines. The one on the far left is known as the lower side-band, the one in the center is the carrier, and the one on the right is the upper-side band.

16. Turn the cursors on and position them until the display appears as follows



17. Compare the theoretical and experimental spectra for the AM signal.

18. Turn the cursors and FFT off (press the Math button again).

FM Frequency Modulation

1. On the FG, produce a 10 MHz sine wave with an rms value of 1 V (1Vrms).
 2. Turn the modulation on (if it is not already) and set it to produce FM (instead of AM) with a Frequency Deviation of 5 MHz and an FM Frequency of 1MHz.
 3. On the scope, press Auto Scale. Observe the display. Reduce the FM frequency to 1 Hz and describe the display (you might need to press Auto Scale again).
-
-

4. Reset the FM Frequency on the FG to 1MHz.
5. Turn on the FFT on the scope and set the Span for 20MHz, the Center for 10MHz and press the Push to Zero button.
6. Manually use the Volts/Div and the vertical position control on the scope to position the waveform on Ch1 on at the top of the display.
7. Set the Time/Div to 1 μ s. Print the display and insert into lab report.
8. Turn the FFT off on the scope and the modulation off on the FG.

Burst

1. Set the function generator to produce a sine wave with a frequency of 10 MHz and an amplitude of 1V_{rms}.
 2. Press Burst on the FG (Fig 1-4) and set the # of Cycles to 5 (button 5th from the left that states “# of Cycles”) and the burst period to 1ms.
 3. Turn the Burst On.
 4. Press Auto Scale on the scope.
 5. On the scope press the spy glass button (Fig 2-17). Vary the setting of the Horizontal knob and position knob to its right and note the effect. Are there 5 cycles?
-
-

6. Turn the spy glass button off and use the Volt/Div and vertical position buttons on the scope to position the waveform near the top of the display. Use the Horizontal knob to change the Time/Div to 5 μ s.
 7. Turn on the FFT by pressing the Math button.
 8. Set the Span to 20MHz, the Center to 10MHz and also press the Push to Zero button.
 9. Turn the cursor on. Use cursors X1 and X2 to determine the width of the main lobe. Print and insert the display into the lab report. How does the width of the main lobe in Hertz compare with the theoretical value?
-
-

10. Turn Burst off on the FG. Turn the FFT and cursors off on the scope.

Lissajous Pattern

1. Make sure the Sync connector on the Function Generator is disconnected.
2. Set the FG to produce a sine wave with a frequency of 1kHz and a peak to peak value of 2V (2V_{pp}) on Ch1.
3. On the FG, turn Ch2 on, set it for High Z, and produce a sine wave with a frequency of 1kHz a peak to peak value of 2V (2V_{pp}).
4. Switch the scope to an XY display. Do this by pressing the Horiz button (it is to the right of the Horizontal knob, see Fig 2-10) and on the display change the Time Mode to XY.
5. On the FG, press Parameters, select phase, and select Sync Internal on one of the channels of the FG.
6. Set the phase on one channel (not both) to 90 degrees.

7. Use the Volts/Div setting and position controls for the two channels on the scope to obtain a display that takes up about 75 % of the display. It should look like an oval.
8. Change the frequency of the Ch1 output on the FG to 2kHz. Press Phase, Synch Internal, and then set the phase to 0 degrees. Set the frequency on Ch2 output of the function generator to 3kHz, the phase to 90 degrees, and then Synch Internal. Print the display and insert in lab report.
9. Set the frequency of the Ch1 output on the FG to 6kHz and the frequency of the Ch2 output to 9kHz.
10. Observe the display for a phase of zero degrees. Print the display and insert into lab report.
11. On the FG, turn the modulation on for Ch1.
12. Select FM with a Frequency deviation of 1Hz and a FM frequency of 0.1Hz. Describe the display.

Computer Automated Measurements

Connecting a computer to laboratory instruments makes possible the automation of many laboratory tasks. Data measurement, processing, storage, and display can be automated. The mechanisms to connect pcs to laboratory instruments are LAN, GPIB, USB, and RS232.

Bode Plots

A Bode plot is a graph of the gain and phase of a two port versus the frequency of excitation. The gain is normally plotted in decibels and the phase in degrees and the frequency in Hertz. The two linear graphs are usually plotted in cascaded windows so that the behavior can be determined at a specific frequency.

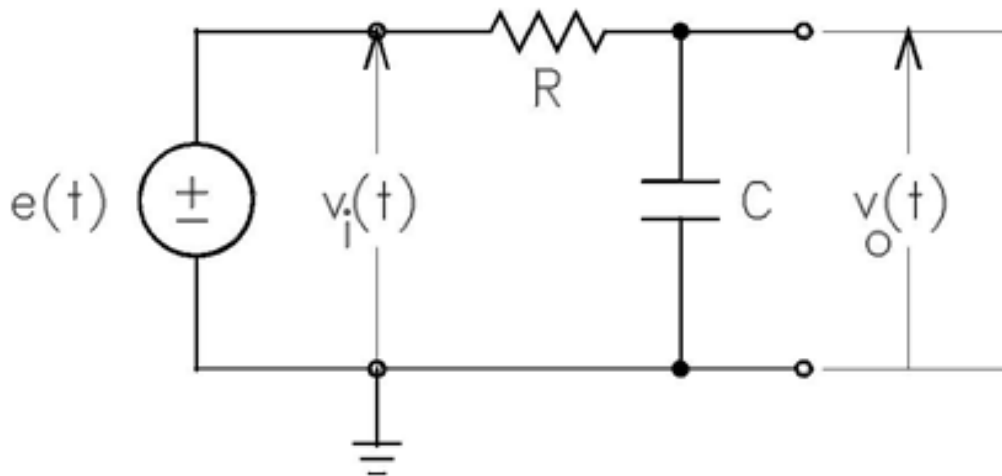


Figure 1. RC LPF Two Port Network

Shown in Fig. 1 is a two port network. The input port is the two terminals across which the input voltage $v_i(t)$ is connected and the output port is the two terminals across which the output voltage $v_o(t)$ is connected. A voltage source $e(t)$ is used as the input. The amplitude of the input source is held constant while the frequency is swept from a start to a stop frequency (for example, from 100Hz to 100kHz). The input and output voltages may be measured with either an AC voltmeter (DMM set to measure AC voltage) or an oscilloscope. A voltmeter can only measure the magnitude of an AC voltmeter; if phase measurements are

required an oscilloscope is necessary. If an oscilloscope is used both of the ground leads must be connected to the ground node shown in Fig. 1.

Magnitude Bode Plot with Agilent FG and DMM using LabVIEW Amplitude Frequency Response

- Assemble the circuit shown in Fig. 1 on **YOUR** breadboard (do not use the ELVIS breadboard). Use $R = 15 \text{ k}\Omega$ and $C = 0.01 \text{ }\mu\text{F}$.
- Connect the Agilent FG to the input [it is $e(t)$] using a coaxial cable BNC to alligator clip lead.
- Connect the DMM (voltage inputs in upper right of instruments) to the output using two banana jack leads.

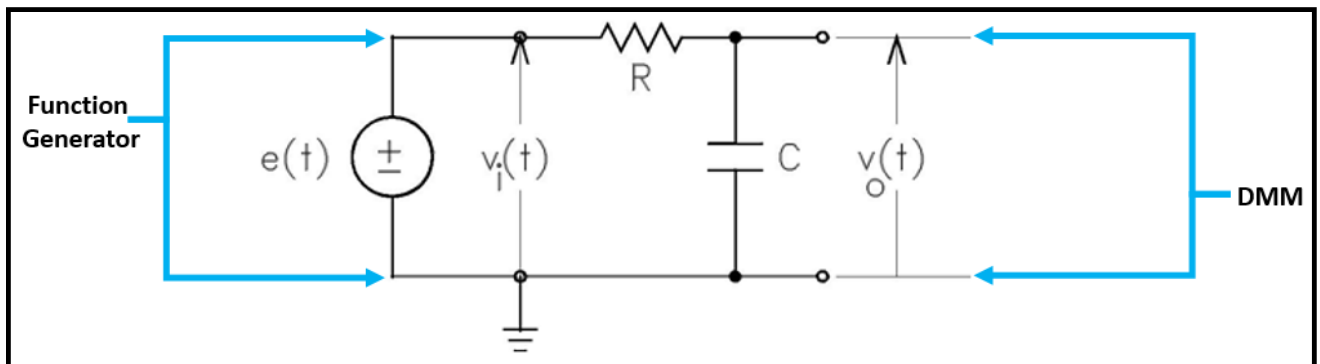
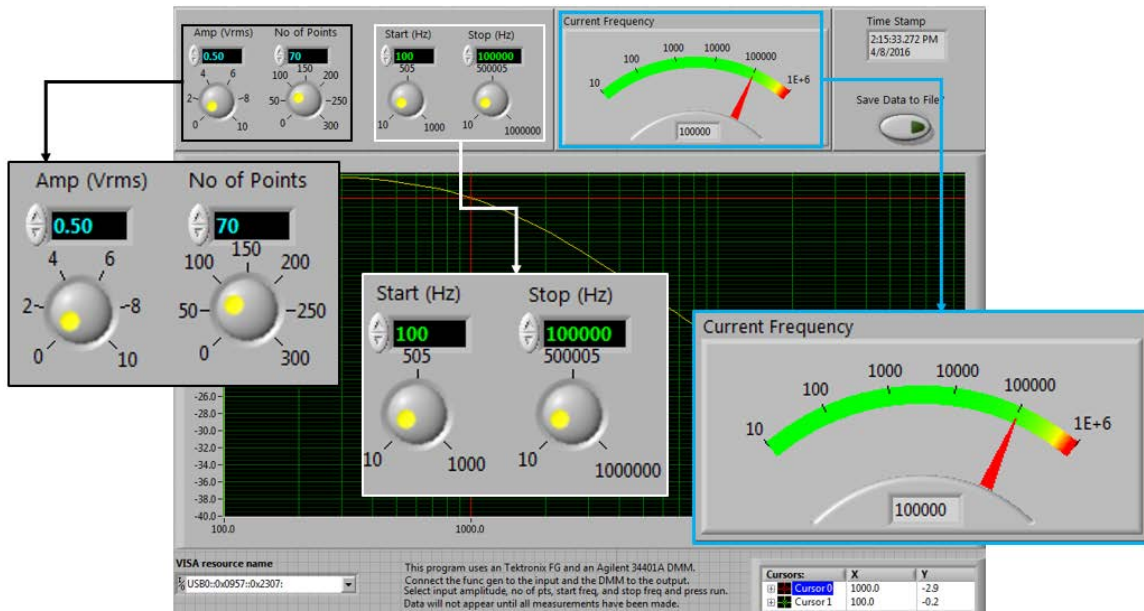
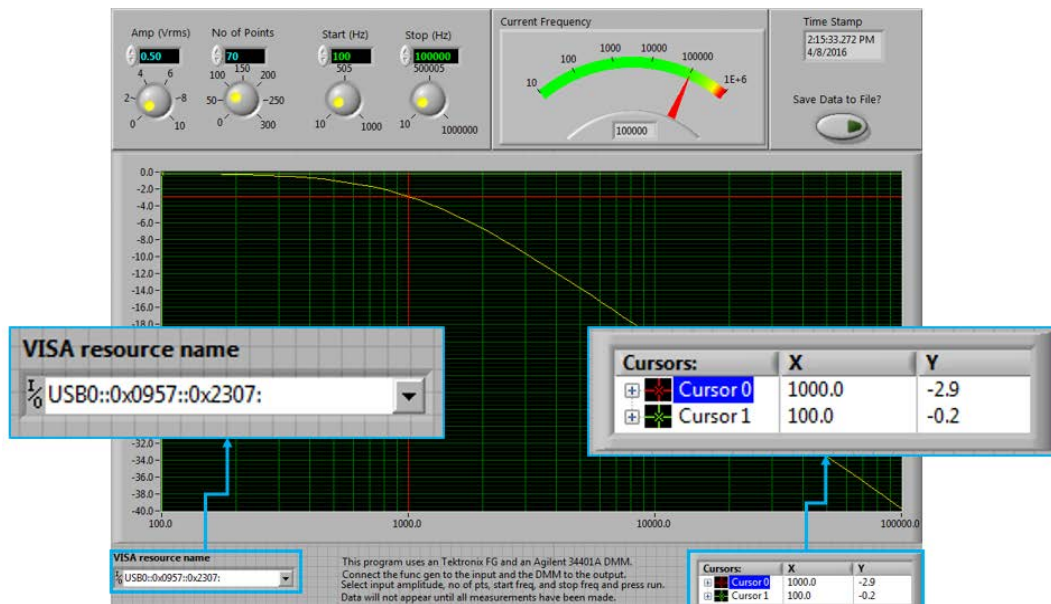


Figure 2. Set up of Function generator and DMM for a circuit.

- Download the program from the class web page known as LabVIEW Amplitude Frequency Response. (If a program called LabVIEW Signal Express starts, close it).
- To the lower left of the generic frequency response plot there is a drop down box **VISA Resource name** to select the location of the FG. Select `USB0::X::X:`
- Set the Start Frequency to 100 Hz and the Stop Frequency to 100 kHz.



- Press Run. After the frequency sweep has concluded (this might take some time depending on the No of points used) use the mouse to place a cursor (a thin red line at the top of the plot that can be dragged down) at -3 dB frequency for the circuit. At the bottom right of the screen there is a table that indicates where the cursor points are.



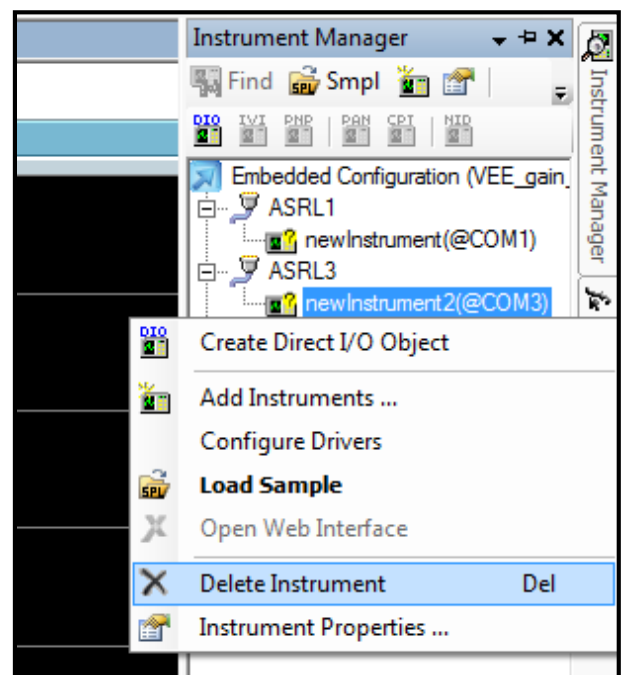
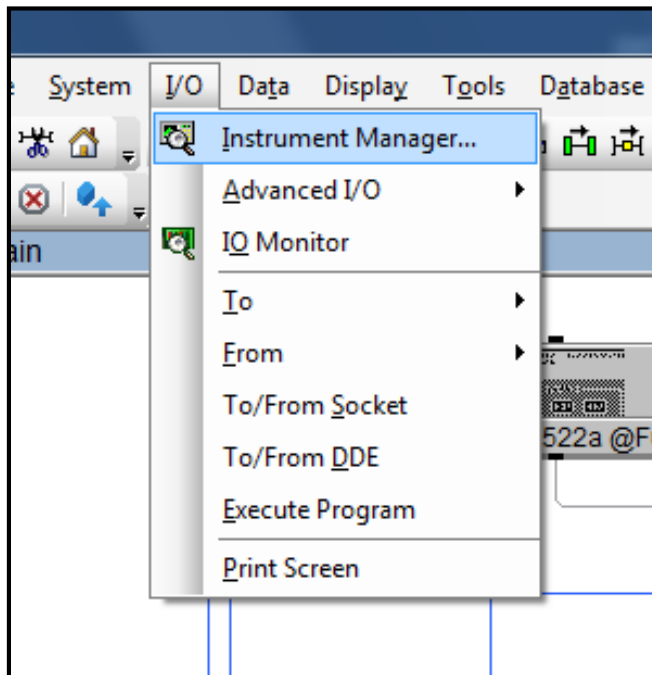
- Press Print Screen and use Paint to place the screen shot into the laboratory report.
- Obtain a Verification Signature.

- Close the LabVIEW program and answer each question with NO.

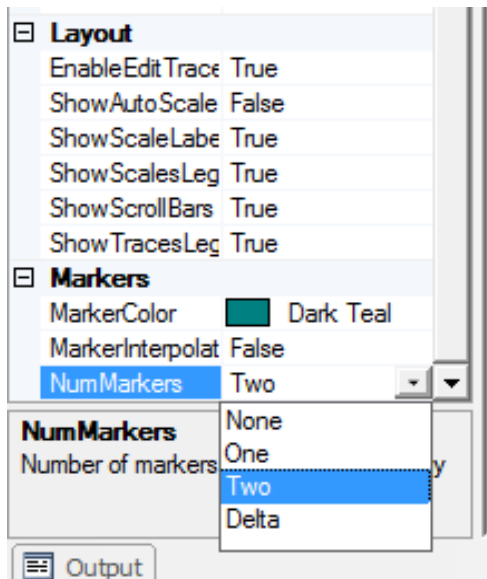
Magnitude Bode Plot with Agilent FG and DMM using VEE Amplitude Frequency Response

- Connect the Agilent FG to the input [it is $e(t)$] using a coaxial cable BNC to alligator clip lead.
- Connect the DMM (voltage inputs in upper right of instruments) to the output using two banana jack leads.
- Download the program called VEE Amplitude Frequency Response from the class web page using “save link as...”
- On the menu click “I/O, instrument manger...”
- Right click and delete all present instrument individually
- Click Find on the menu
- Click OK as many times as you are prompted
- Right click on the instrument, instrument properties, and **only change the name** (don’t change other fields) the instruments as shown in chart below:

Name	Instrument
FuncGen	ag33522a
DMM	4hp34401



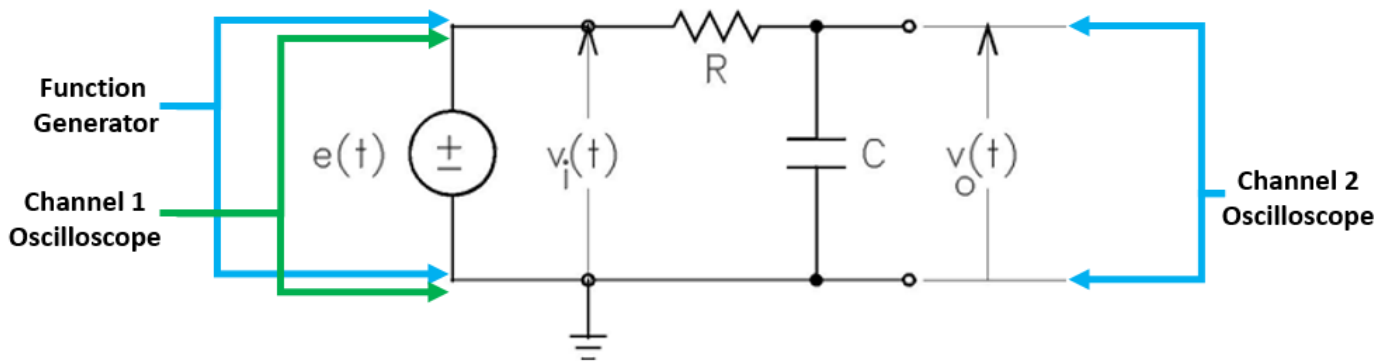
- Do not change frequency range.
- Press run (green arrow on top of screen).
- Right click on the plot, go to properties, these will be on the left side of the screen, scroll down to markers, and use the drop down box to select one cursor and turn the other parameter on.



- Position a cursor at the minus 3 dB frequency.
- Do a print screen and use Paint to put the Screen shot into the lab report.
- Obtain a Verification Signature and close the program.
- Remove the connections from the circuit to the DMM.

Magnitude Bode and Phase Plots with Agilent FG and Agilent Scope using VEE Amplitude Phase Frequency Response

- Connect the input of the circuit to CH1 of the scope using BNC to alligator clips coaxial leads.
- Connect the output of the circuit to CH2 of the scope using BNC to alligator clips coaxial leads.
- The function generator is (still) connected to the input.



- Download the program VEE Amplitude Phase Frequency Response from the class web page using “save link as...”
- On the menu click “I/O, instrument manger...”
- Right click and delete all present instrument individually
- Click Find on the menu
- Click OK as many times as you are prompted
- Right click on the instrument, instrument properties, and **only change the name** (don’t change other fields) of the instruments as shown in chart below:

Name	Instrument
FuncGen	ag33522a
scope	agdso

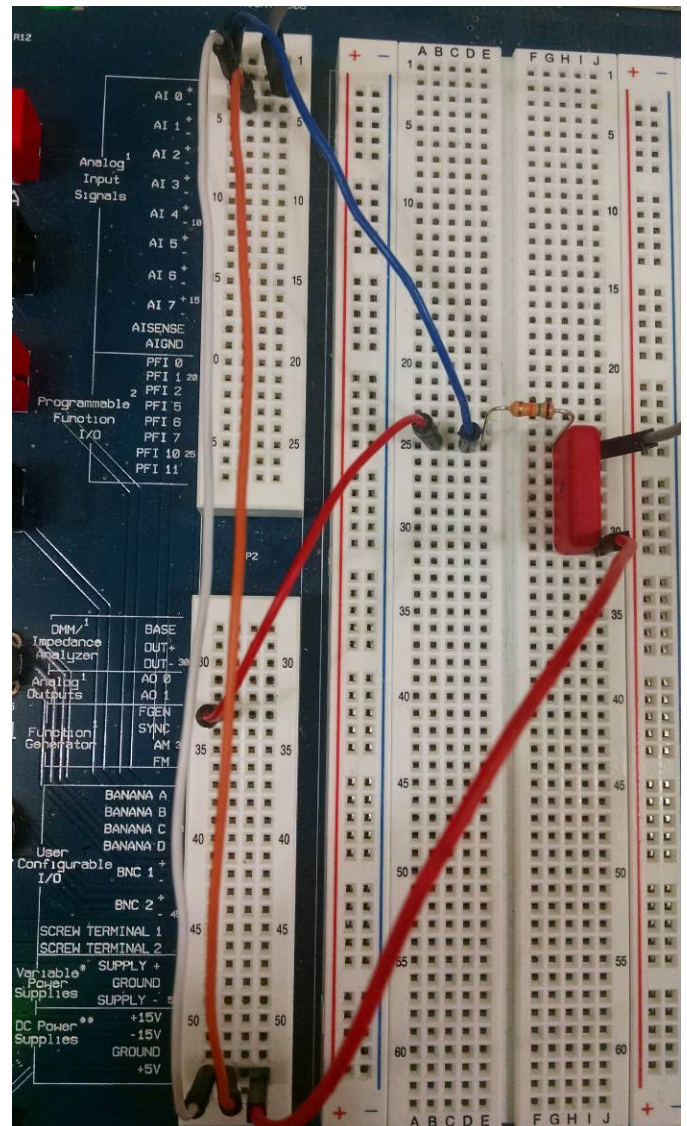
- Press Run.
- Click check box to turn cursor on.
- Place a cursor at the minus 3 dB frequency and use Print Screen and Paint to place the result in the lab report.
- Disconnect all leads connected to the function generator, the DMM, and the scope.

Magnitude Bode and Phase Plots with ELVIS

ELVIS can also be used to obtain both magnitude and phase Bode plots. The ELVIS function generator provides the input. The analog inputs 1 and 2 measure the input and output of the circuit respectively.

- Transfer the first-order circuit from your breadboard to the ELVIS breadboard.

- Turn on the ELVIS board (switch on back and top).
- Start the National Instruments Launcher on the PC. Start Programs, All Programs, National Instruments, NI ELVISmx for NI ELVIS & NI myDAQ, NI ELVISmx Instruments Launcher, Bode.
- On the Measurement Settings change the Stimulus Chanel to AI0 and the Response Chanel to AI1.
- **The following ELVIS connections are on the breadboard section on the far left;** they are difficult to read; position yourself directly over the breadboard when making these connections.
- Connect AI0- and AI1- analog input signals to the DC power Supply ground located on the lower left side of panel next to the breadboard (the vertical trace on the ELVIS breadboard is broken half way down so a jumper will be necessary) with jumper wires.
- Connect the ground of the circuit to the DC power Supply ground
- On the ELVIS board, connect the AI1+ analog input signal pin to the output of the circuit.
- Connect the ELVIS Function Generator FGEN pin and AI0+ analog input signal pin to the input of the circuit.
- Press Run on the ELVIS front panel on the screen of the computer.
- Turn on the cursors and position them to the half power frequency.
- Press Print Screen and use Paint to paste the result into the laboratory report.
- Remove all the jumper wires and components from ELVIS and turn off.



Voltage Transfer Characteristic

A Voltage Transfer Characteristic plots the output voltage as a function of the amplitude of the input voltage. For a passive circuit this would be a straight line. It is meaningful only if the circuit has a nonlinear component such as a diode, transistor or integrated circuit.

For the circuits specified by the Laboratory Instructor obtain the Voltage Transfer Characteristic (VTC).

- Typically, this would be a resistor in series with a diode. The input goes across the series combination and the output is taken from the junction between the diode and resistor.
- The Voltage Transfer Characteristic program is found under Software Applications on the class web page.
- After it opens there are three tabs. Click on the Devices tab
- Run it (press the white arrow) and it should detect ELVIS.
- Click on the Diodepara82 tab and build the circuit as shown. Use a 5.1k Ω resistor and 4148 Diode.
- After the VTC has been obtained for each circuit do a print screen and use paint to obtain a sketch that can be pasted into the lab report.

Arbitrary Waveform Generator

Perform the procedure on the main class web page titled Arbitrary Waveform Generator.

Programming Arbitrary Waveforms into the Waveform Generator

Procedures:

- Use Mathcad to create the data for the FG. The below is a template for a specific function consisting of the sum of two sine waves.
- Save the data generated in a text file somewhere using Notepad.
- Turn the Function Generator on.
- If Signal Express starts on the computer, close it. The waveform to be programmed into the Function Generator will be specified by the laboratory instructor.
- Download the LabVIEW program called Arbitrary Waveform Generator from the Software Applications section of the class web page.
- In the upper left hand corner of the LabVIEW front panel will be a control called VISA resource name.
- Click the selection arrow in this slot and click the address of the USB0 device.
- Set the Output Amplitude to the desired value on the front panel.
- Click the Run arrow at the top of the front panel.
- The program will then ask for the location of the data file. Enter it.
- The display on the FG should now indicate a picture of the waveform entered.
- Turn on the oscilloscope.
- Connect the output of the FG to the oscilloscope.
- On the FG press Local (Under System Button).
- Press Sample Rate and then enter 9 MS/s.
- Press Auto Scale on the oscilloscope. Print the screen.
- Program any additional waveforms specified by the laboratory instructor.

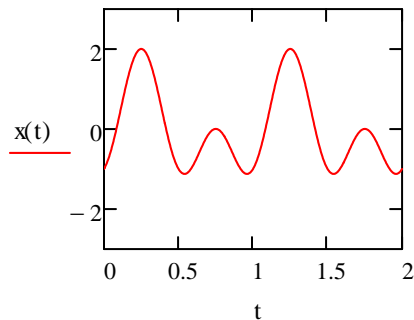
Programming the Function Generator for Two Tones

$$f_p := 1 \quad \omega_p := 2\pi \cdot f_p \quad x(t) := \sin(\omega_p \cdot t) + \sin\left[2 \cdot \omega_p \cdot t - \frac{\pi}{2}\right]$$

Sample with 16000 points for the Function generator:

$$N := 16000 \quad i := 0..N-1 \quad d_i := x\left(\frac{i}{N}\right) \quad \text{Max} := \max(d) \quad \text{Min} := \min(d)$$

$$\text{Max} = 2 \quad \text{Min} = -1.125$$



	0
0	-0.92
1	-0.92
2	-0.919
3	-0.919
4	-0.919
5	-0.919
6	-0.918
7	-0.918
8	-0.918
9	-0.918
10	-0.917
11	-0.917
12	-0.917
13	-0.917
14	-0.916
15	...

Normalize points for Function Generator from -1 to +1

$$d_i := \left[d_i - \frac{(\text{Max} + \text{Min})}{2} \right] \cdot \frac{2}{\text{Max} - \text{Min}}$$

d =

To save data on notepad:

- Position pointer in array, right click, select all (if it's not already selected), copy selection
- Open Notepad, right click, paste.
- Save your file, you will need it later.

Notes

Shortcut for f_p is a dot (p is a subscript) ex: f.p

In $i := 0..N-1$ the shortcut the two dots following the zero is a semicolon ex: 0;N-1

Shortcut for d_i is the left square braquet (i is an array index) ex: d[i

To make the table just write d= and press enter.