Laboratory 1: Lab Equipment Introduction

Introduction:

0.1 Overview

This is the first of a series of labs that you will work on through out the quarter. Each lab will contain three parts: a prelab, a computer simulation, and an experiment. The prelab uses theoretical aspects to design and analyze circuits. The simulation part of each lab will focus on implementing the circuits from the prelab in a circuit simulator (OrCAD) which provides an imitation of the response of circuits before the actual implementation. With OrCAD, the circuit analysis and design will be verified using ideal circumstances. The final part of the lab will be experimentation. For this section, the designed circuit will be assembled on a breadboard and the circuit's performance will be compared to the results from the prelab and simulation.

For the first lab, you will be introduced to the four Keysight instruments and explore their various functions. There is the digital multimeter (DMM), oscilloscope, function (or waveform) generator, and triple-output DC power supply. You will also learn how to use OrCAD to run simulations of a circuit.

0.2 Preparation

0.2.1 Intro to Lab Equipment

To help introduce you with the 4 main electronic instruments in this class:

Watch before you come to lab:
Required Watching 1
Required Watching 2
Required Watching 3

0.2.2 How to use a Breadboard

A solder-less breadboard is a reusable construction base for prototyping circuits. It has two power rails and terminal strips, which is separated by a gap in the middle. This gap is used to support dual in-line package (DIP) components, which have rectangular housing and two parallel rows of electrical connecting pins. See Figure 1 for an example of the internal structure of a breadboard.

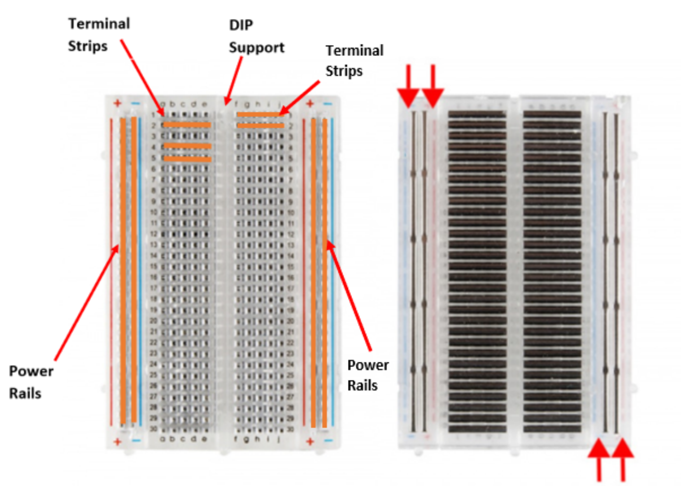


Figure 1: Internal structure of a breadboard

a. Power rails are located on sides of your breadboard. When plugging power to the '+' rail and ground to the '-' rail, the entire column of the '+' rail will have the same voltage as power and the entire column of the '-' rail will have 0V. Note: the power rails on either side of the breadboard are not connected.

b. DIP support is designed specifically for you to insert the chips. See Figure 2 as an example of how DIP support is used. Note that the power strips on either side of DIP support is not connected.

c. Terminal Strips are rows on the breadboard that has the same voltage.

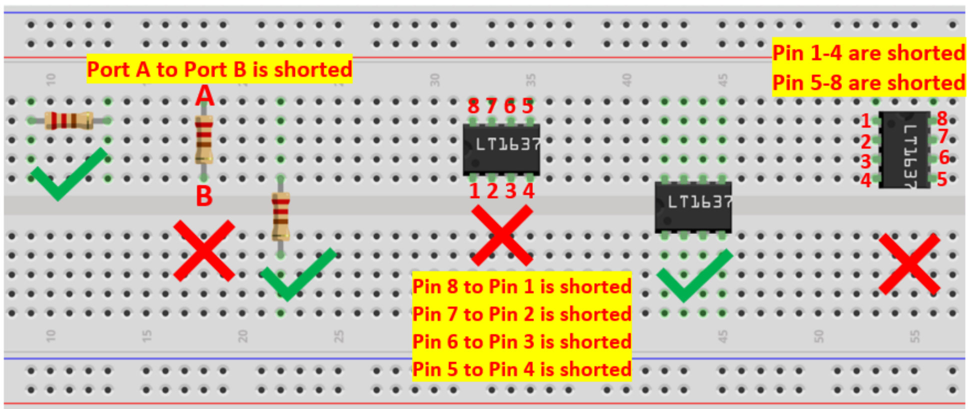


Figure 2: How to correctly place chips and resistors onto breadboard

1 Prelab

1.1 Resistive Network

For the circuit in Figure 3

1. Calculate the current, voltage potential, and power for each resistor, and fill out Table 1. Assume $V_{in} = 5V$

Resistor	Resistance (Ω)	Voltage (V)	Current (mA)	Power (W)
R_1	1k			
R_2	10			
R_3	10			
R_4	34			
R_5	1k			

Table 1: Results of circuit in Figure 3

2. What is the equivalent resistance?

$R_{eq} =$ ____________

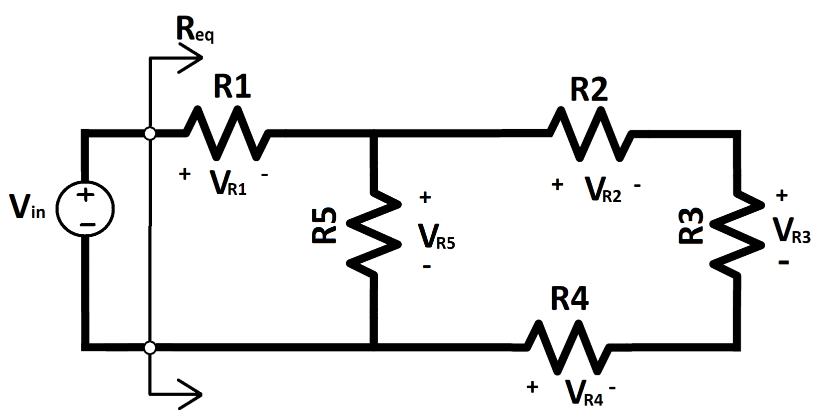


Figure 3: Resistive Network

2 Simulation

We use Cadence OrCad to simulate our circuits before we build them on breadboard. Please adhere to the following procedures carefully. If you have a question, ask the teaching assistants for assistance. Several videos tutorials of OrCad/PSpice are also available online if one wishes to obtain more information on using the software

2.1 Resistive Network

1. Launch “Capture CIS Lite” software from the windows menu.
2. Create a new project
   (a) Select File > New > Project....
   (b) Enter “Lab1 Series Network in the Name input field in the New Project dialog.
   (c) Select the “Analog or Mixed A/D” option.
   (d) Specify where the project will be saved in the Location field by clicking on the Browse... button.
   (e) Create a folder “Lab1” and save your project there.
   (f) Click OK to confirm project creation.
   (g) Choose “Create a blank project” in the Create PSpice Project dialog and then click OK. NOTE: A blank schematic sheet with tab title PAGE1 will appear.

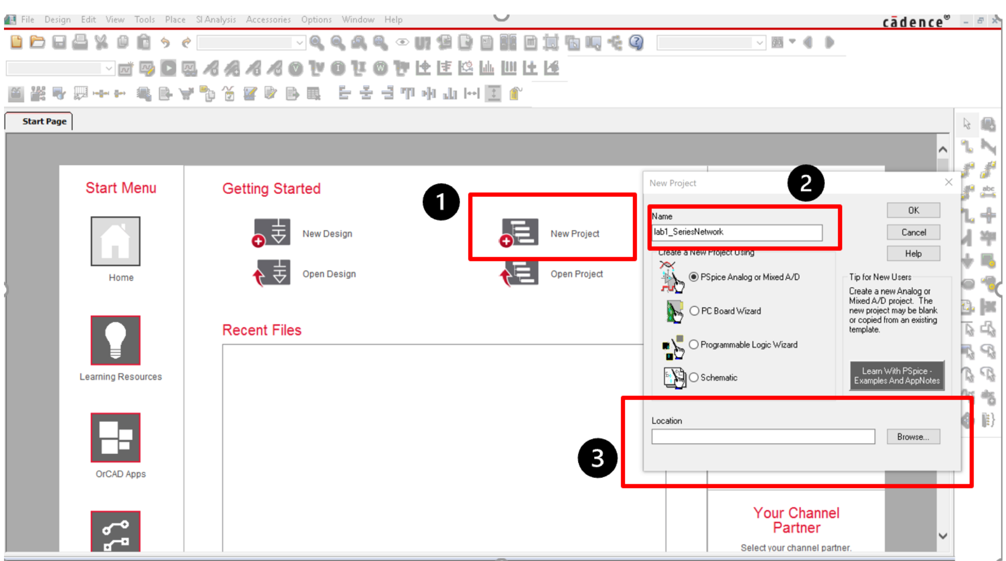
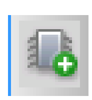
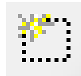


Figure 4: Creating new project on Cadence

3. Hit P on the keyboard or click  icon located on the right-hand panel to open the Place Part panel.
4. Use the drop-down arrow of the Browse File dialog to ensure the file path is:

```
C:\Cadence\SPB_XX.X\tools\capture\library\pspice
```

Note: XX.X is the version number of Cadence. The version should be ≥ 17.2.

5. Click the Add Library button  in the Place Part panel, under Libraries.

6. Hold ctrl on your keyboard and select all of the following files at the same time:

   - analog.olb
   - breakout.olb
   - source.olb
     Note: Only components from the library (.olb files) can be simulated by Pspice.

7. Click Open to confirm selection.

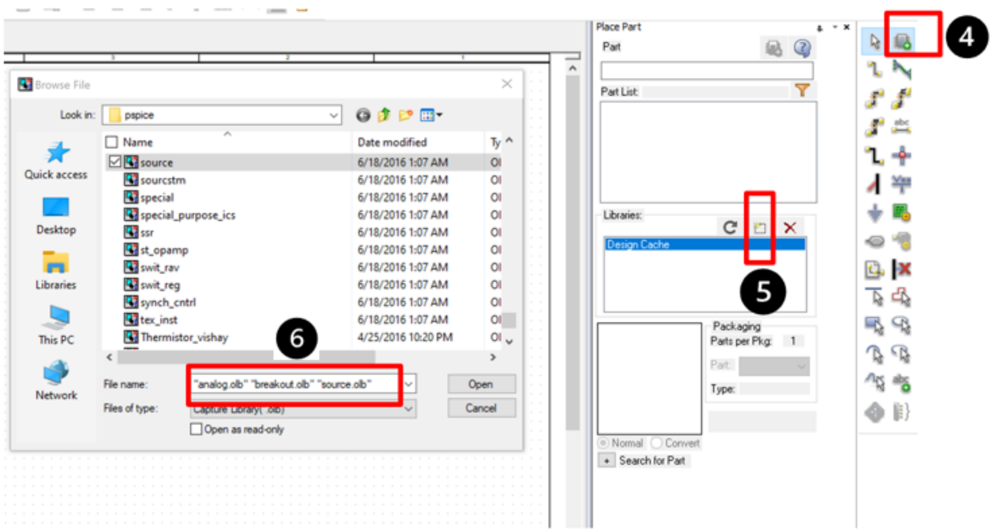


Figure 5: Import parts into library

8. Select "ANALOG" from the library menu.

9. Double click 'R' for resistor in the parts list menu.

10. Place the resistor onto PAGE1.

    (a) Left-click once on PAGE1 canvas to place the resistor.
    Note: To exit the part placement, press ESC. OrCAD will continue to place resistors based on the location and mouse clicks (i.e. if you left-click 3 times on PAGE1, OrCAD will place 3 resistors until you hit ESC).

    (b) Double click on the "1k" located beneath the resistor to bring up the Display Properties.

    (c) Change the value of the resistor by entering the R1 resistance value in the Value box.

    (d) Select "Value Only."

    (e) Click OK when finished.

11. Repeat steps 10a-e to place and set the resistance for resistors R2, R3, R4, and R5.

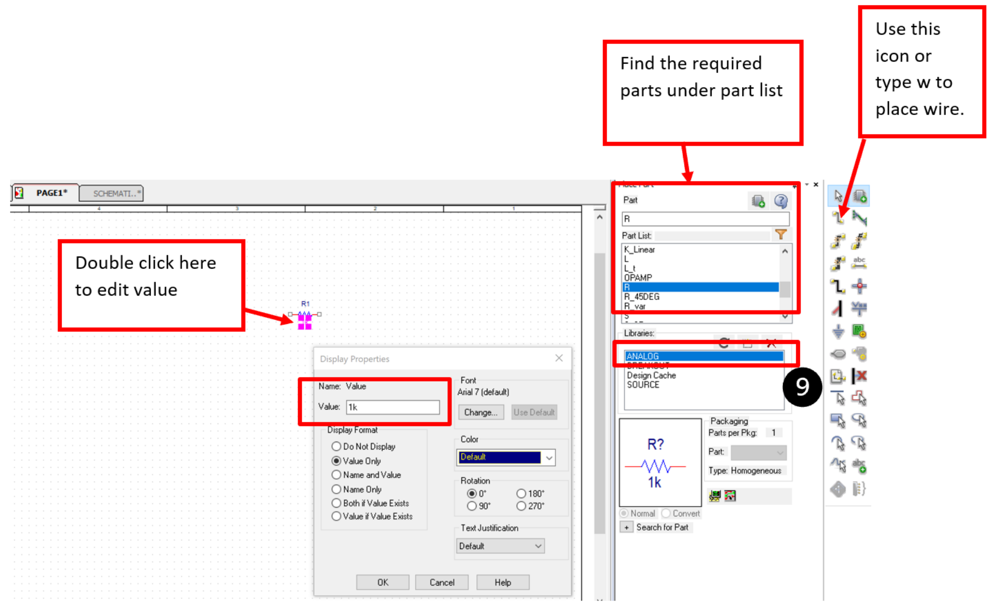
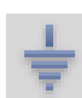
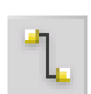


Figure 6: Place parts and build circuit scheme

12. Select "Source" from the library menu.
13. Search and place "Vdc" from the Place Part panel onto PAGE1.
14. Double click the "0Vdc" to bring up the "Display Properties" tab.
15. Enter "5Vdc" for the "DC" parameter to set this DC source +5V.
16. Select "Value Only."
17. Click OK to apply changes.
18. Select the ground icon  on the right-hand panel.
19. Select "0/CAPSYM" in the Place Ground dialog to ensure ground has 0 potential.
20. Click OK to save changes.
21. Place ground onto PAGE1.
22. Hit w on your keyboard or click  to wire and connect the terminals of each part so it resembles Figure 7.

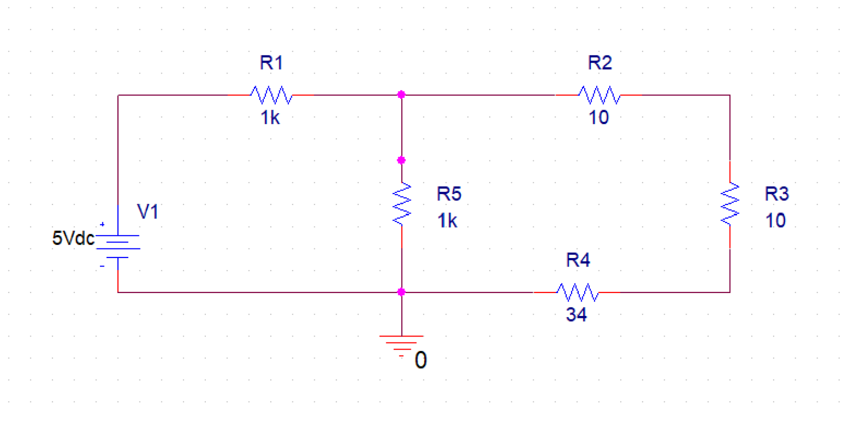


Figure 7: Resistive Network Schematic.

23. Click PSpice > New Simulation Profile from the menu bar on the top of the screen.
24. Enter "sim1" in the Name field and leave the remaining to the default value.
25. Click Create.
26. Under the Analysis tab, choose "Bias Point" as the Analysis Type in the new pop-up window, Simulation Settings.
27. Click OK.

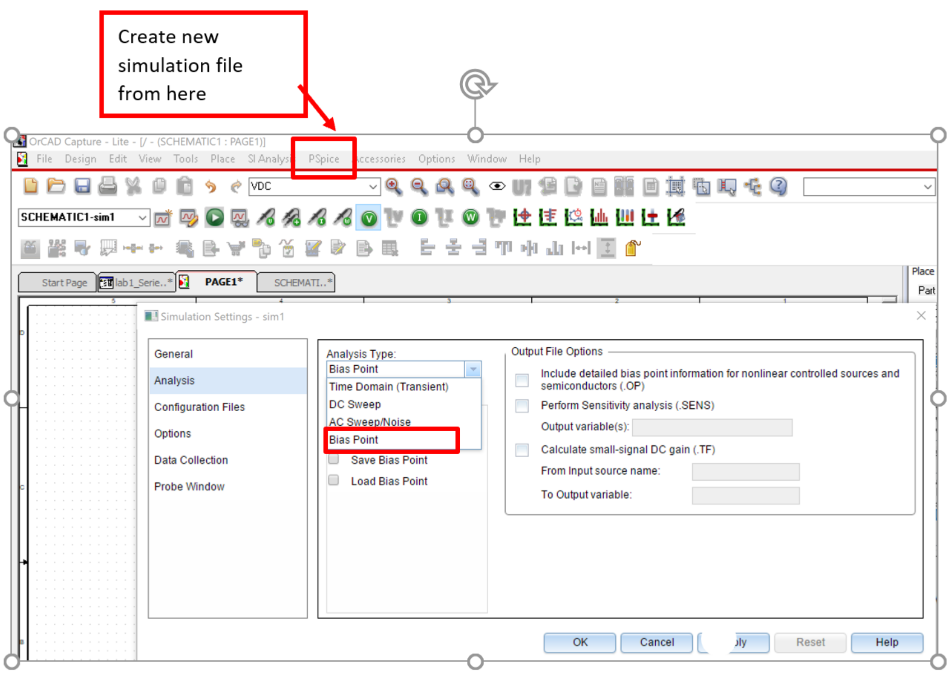


Figure 8: New Simulation Profile Settings

28. Run the simulation by clicking on the  icon to bring up the "PSpice A/D" window.

29. Press the , , and  buttons to display the voltage, current, and power values of the circuit.

30. Fill out the following table.

	Voltage	Current	Power
R_1			
R_2			
R_3			
R_4			
R_5			

31. Calculate the equivalent resistance. (Hint: Use the current flowing through the voltage source).

R_{eq} = ____________

3 Experimentation

3.1 Oscilloscope Control Settings

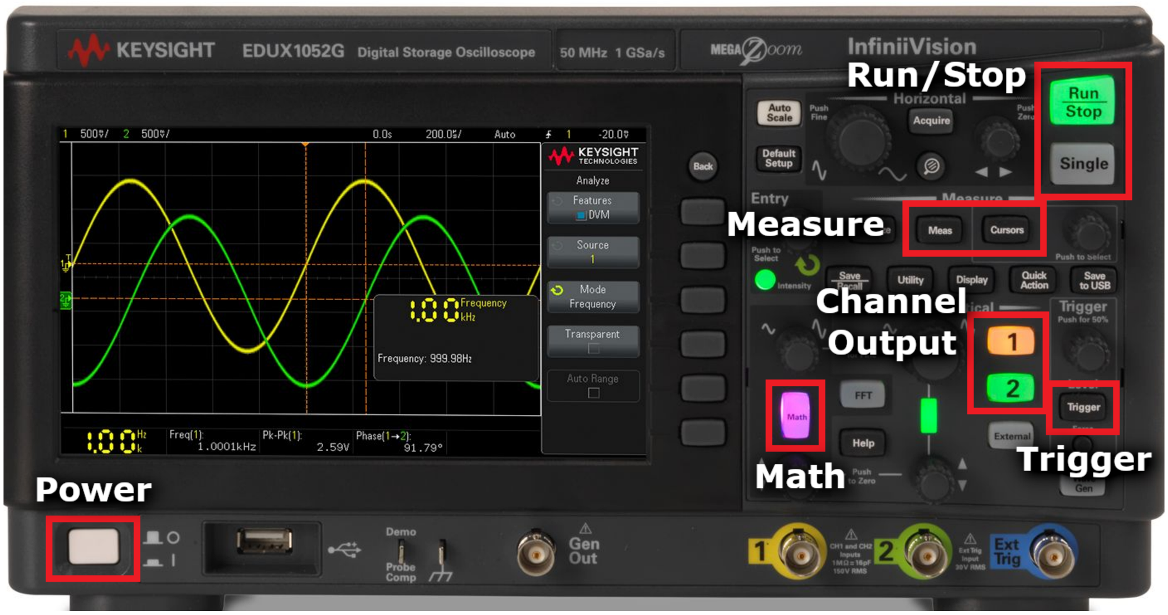


Figure 9: Oscilloscope Main Buttons

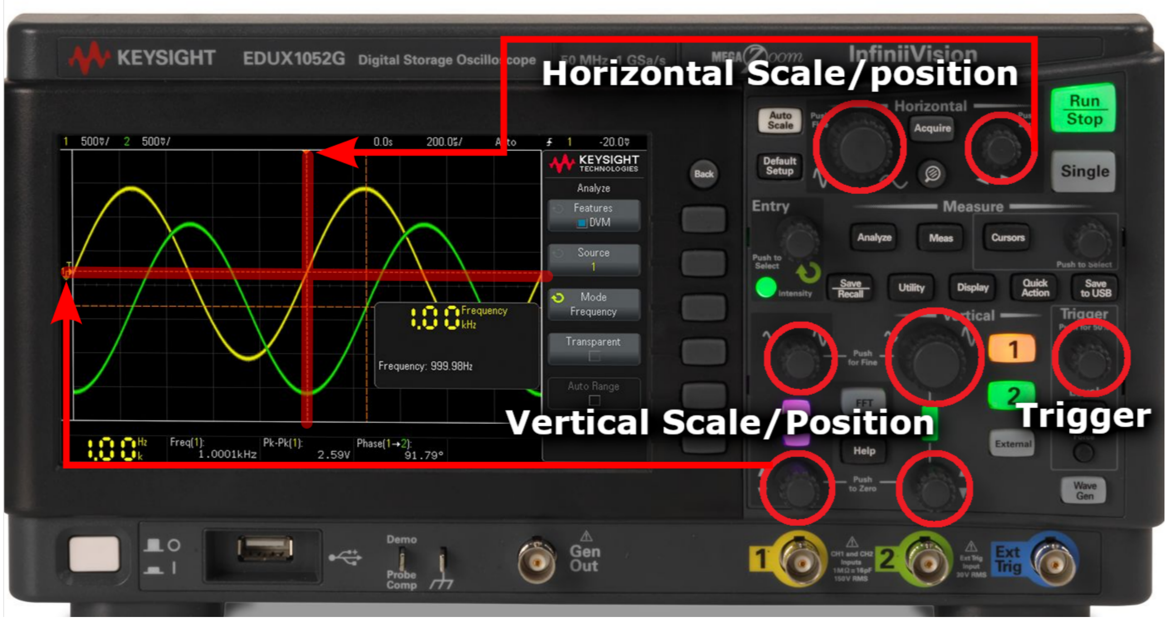


Figure 10: Oscilloscope Main Knobs

An oscilloscope is a device for viewing voltage oscillations on a display. The oscilloscope reticule is the lined display grid. The vertical axis of the display represents signal amplitude (voltage), while the horizontal axis represents time.

1. Turn the power/line switch **ON** in the bottom left corner of the scope.
2. Enable channels 1 and 2 of the Scope by pressing the [**1**] and [**2**] channel output buttons so they light up.
3. Set the vertical scale for each channel to **500 mV/Div** using the big knobs and the upper left corner of the screen.

Figure 11

4. Set the horizontal scale (bigger knob) to 10 ms/div.

Figure 12

5. Press the [**Trigger**] button (see Figure 9). Configure the **Trigger Mode** to Normal, the **Trigger Type** to Edge, the Channel Source to 1, and the **Slope** to Rising ( ). Finally, set the **Trigger Level** to 0 V by pressing the **Trigger** knob (see Figure 10).

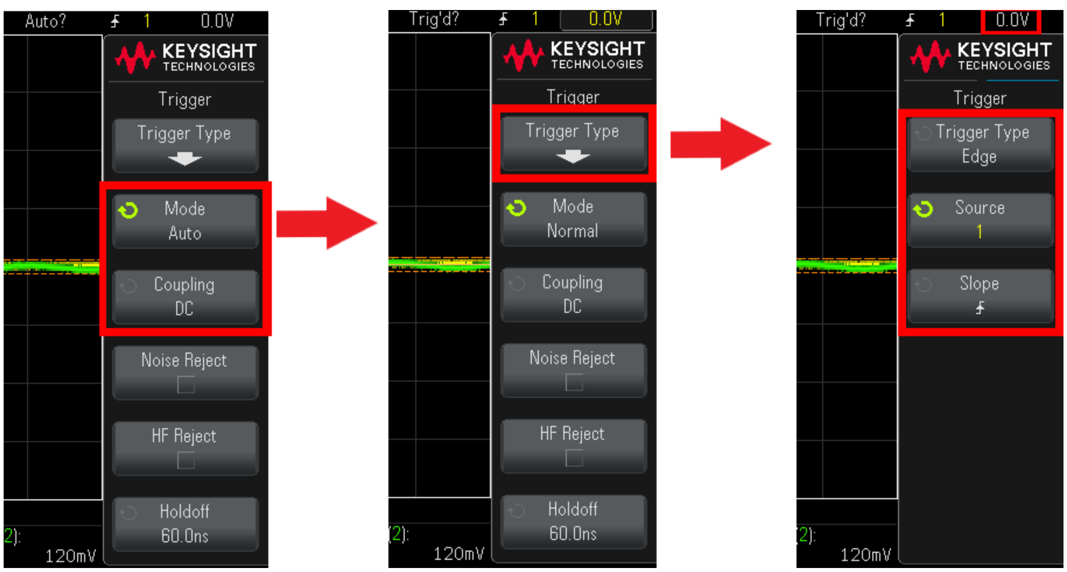


Figure 13

6. Disable both channels by pressing the [**1**] and [**2**] buttons until they are no longer lit up and the waves on the screen disappear.
7. Enable Channel 1 (by pressing the [**1**] button until it lights up yellow); move the trace **up** and **down** by adjusting the knob under the [**1**] button, center it by pressing the knob, then turn Channel 1 **off**.

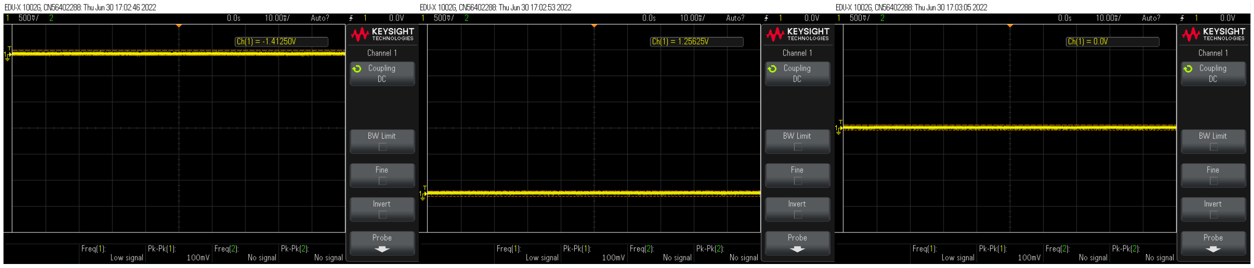


Figure 14

8. Enable Channel 2 (by pressing the [2] button until it lights up green); move the trace **up** and **down** by adjusting the knob under the [2] button, center it by pressing the knob, then turn Channel 2 **off**.

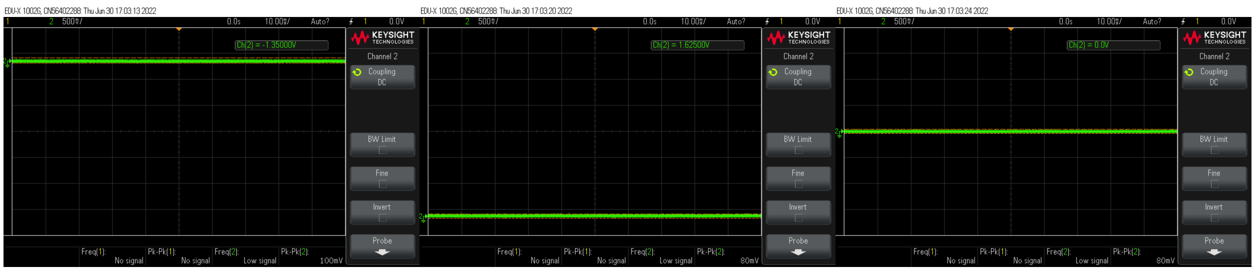


Figure 15

9. Enable Channel **1** again; you will use Channel **1** for the rest of the lab.
10. Adjust the Channel 1 Probe Attenuation to **1.00 : 1**.

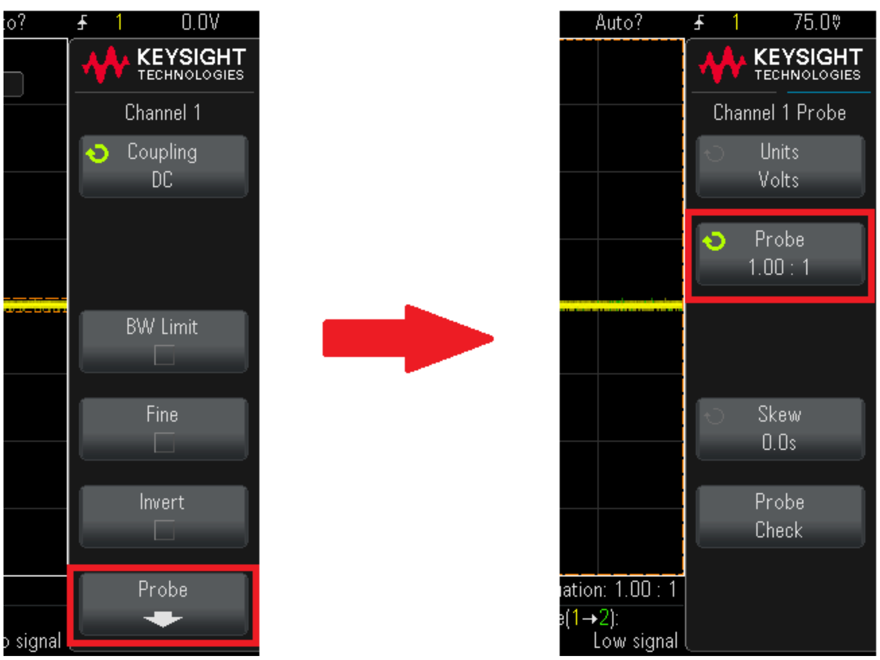
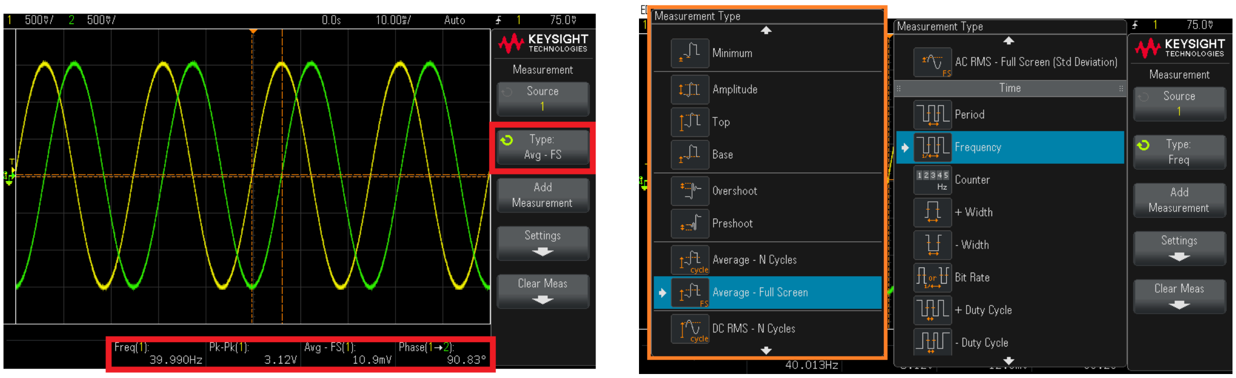


Figure 16

11. To configure measurements on the two signals, press the [**Meas**] button to choose which measurements to perform on the waveforms acquired by the scope. You can have up to **4** measurements displayed at the same time. Press **Clear Meas** to remove the measurement you want to replace, otherwise the left-most measurement will be replaced. Select **Type** to choose the kind of measurement you want to add.

Example measurements:



(a) (b)

Figure 17: (a) Four measurements on the scope screen. (b) Adding measurements to the screen.

You should NOT trust the function generator's display and instead use the scope or a DMM to ensure you have the correct voltage output.

3.2 Function Generator Control

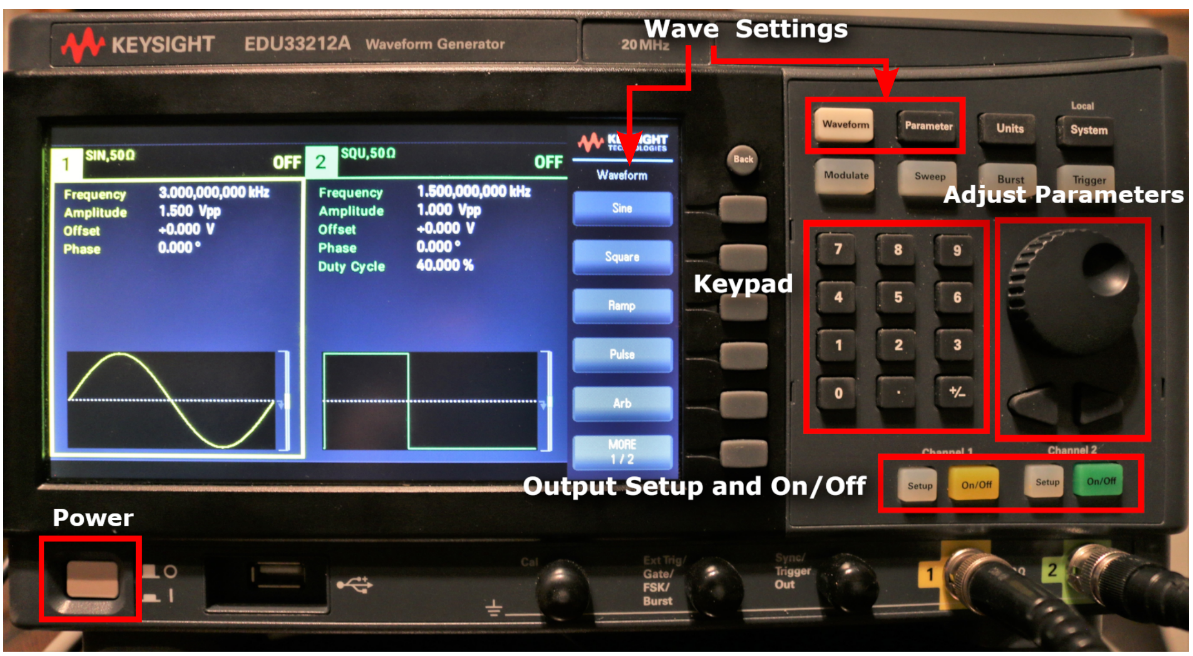


Figure 18: Function Generator Main Controls

1. Turn **on** the power if not already on.
2. Press the [**Setup**] button for Channel 1. Then press [**Waveform**] and select **Sine** on the screen.

Figure 19

3. Press the [**Parameter**] button.

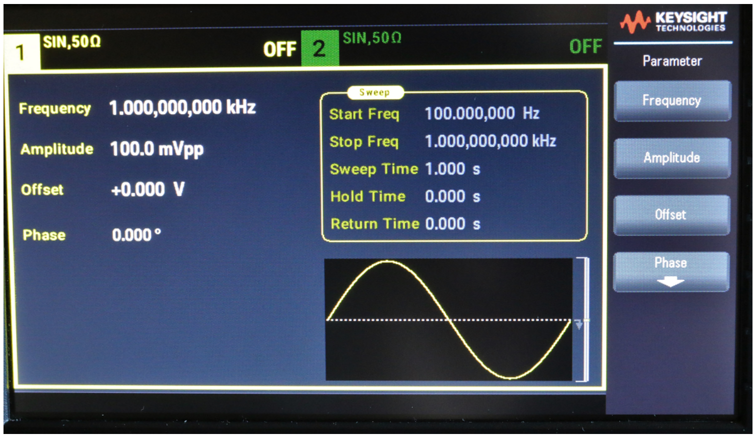


Figure 20

Note: The parameter values on the function generator can be set by selecting the parameter you want to change and then either (1) typing the number on the **Keypad** and then selecting the correct units button (right side of the screen), or (2) using the large scroll knob and the left/right arrows under the knob.

4. Set the amplitude to 0.2 Vpp (type ".2" and select **Vpp**).

Figure 21

5. Set the offset to **0 V DC** and the Phase to **0°**.
6. Set the frequency to **100Hz**:

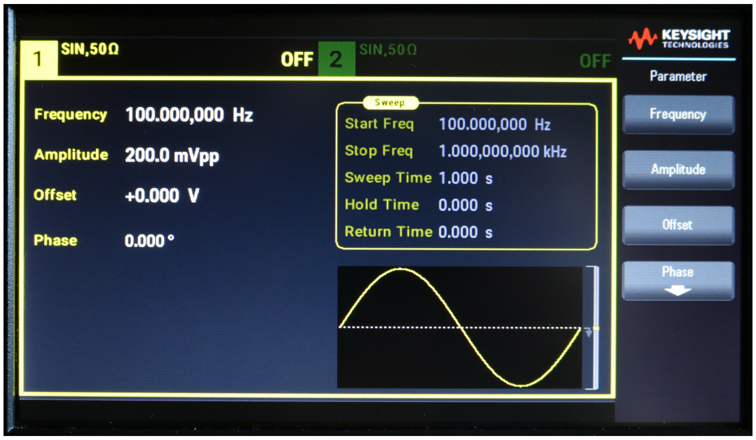
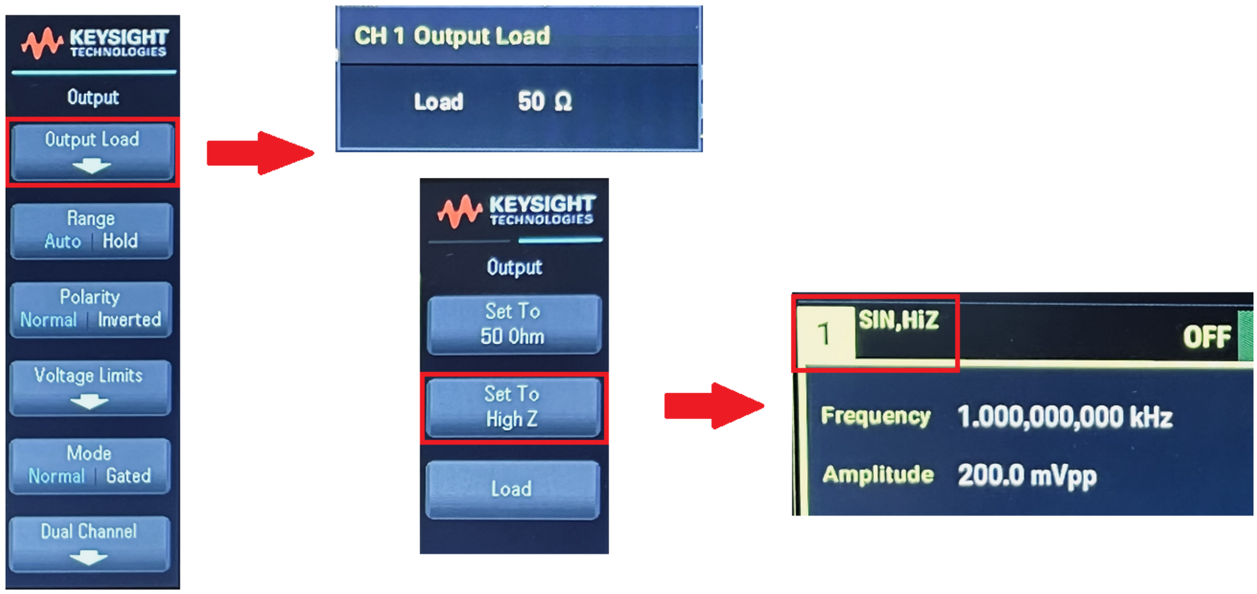


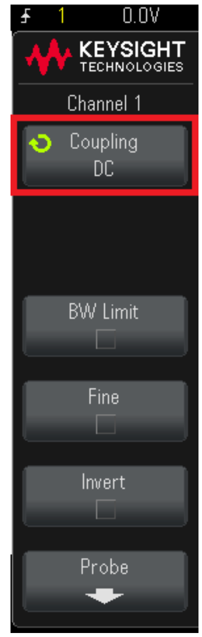
Figure 22

7. For the oscilloscope and the function generator to agree on the voltage of the signals being generated, the load impedance must match the internal resistance of the oscilloscope (≊10 MΩ). Press the [**Setup**] button for Channel 1. Select **Output Load** and set the output impedance to **High Z**, as shown below.



8. Connect the function generator **OUTPUT** to the scope's Channel **1** input using a single BNC-to-BNC cable (These have circular ends). Make sure the probe attenuation is still set to **1.00 : 1**.
9. Switch **ON** the output of the function generator.
10. Adjust Channel 1's **Volts/Div** on the oscilloscope so the waveform is as large as possible, but within the top and bottom horizontal reticule lines.
11. Adjust the **Time/Div** control so you see **two periods** of the waveform on the screen.
12. Adjust Channel 1's **position** so the bottom of the waveform is on **the bottom** reticule line.
13. Take a screenshot of this wave filling the oscilloscope screen.

3.3 AC Voltage Measurement Using the Scope



1. Keeping the results from the previous section and leaving the Signal Generator and Oscilloscope on, ensure the Amplitude of the function generator is **0.2 Vpp**.
2. On the oscilloscope, ensure channel 1 is DC coupled (to allow both AC signal and DC signals to be observed) by pressing the [1] button to display channel 1 settings and change Coupling to **DC**, as shown to the right.
3. Set the Trigger mode to **Auto**. Set the trigger level to **300 mV**. The waveform will be unstable and moving all over the place. Decrease the trigger level slowly until the wave is stable (not moving). Record the highest trigger level that still lets the wave be stable.

   ____________ V_{pp}

4. Set the trigger level to **−300 mV**. The waveform will be unstable and moving all over the place. Increase the trigger level slowly until the wave is stable (not moving). Record the lowest trigger level that still lets the wave be stable.
5. Set the Trigger mode to **Normal**. Adjust **trigger level** and the **horizontal position** so that one of the sine wave peaks is centered on one of the major vertical lines near the center.

   Note: In Normal mode, the waveform you see is out-of-date if you see "**Trig'd?**" (with question mark) near the top of the screen. In Normal mode, the waveform is only accurate while the scope is running (green Run/Stop button, not red) and you see "**Trig'd**" (without the question mark).

6. Record the Volts/Div setting for channel 1 on the top-left of the display:

   ____________ mV/Div

7. Estimate the peak-to-peak height of the waveform to the nearest tenth of a division (Hint: use the subdivision ticks). For accuracy, take the waveform trace (line) thickness into account:

   ____________ Div

8. The peak-to-peak value of the waveform (in volts) is the division difference between the peak and valley of the waveform times the scope's sensitivity in Volts/Div. Calculate the peak-to-peak voltage.

   ____________ V_{pp}

9. Compare the value you calculate with the value reported by the scope's automatic measurements (Add a Voltage - **Peak-to-Peak** measurement on Channel 1, see Figure 17b).

   ____________ V_{pp}

10. The standard specification of a waveform's amplitude is its zero-to-peak (or simply peak) voltage. For a sinusoid, this is one-half the peak-to-peak voltage. Convert the peak-to-peak voltage amplitude Vpp in part (10) to peak voltage, Vp.

    ____________ V_p

11. Compare the value you read with the value reported by the automatic measurements (Add a Voltage - **Top** measurement on Channel 1).

    ____________ V_p

3.4 Period and Frequency Measurement Using the Scope

The oscilloscope can be used to measure **a signal's period** (the duration of one cycle of the signal) and the frequency (number of cycles per second). These are both measured with the horizontal (time) scale of the scope display. The space between the bigger vertical lines on the scope graticule (the display grid) is considered to be one "division". This space corresponds to the time-base setting (how fast the signal is drawn across one division). The time between two points on the signal can be found by multiplying the horizontal distance between the points (in divisions) and by the time-base setting (in seconds/division).

1. If the generator's frequency is still 100 Hz, then there should be two cycles (periods) on the screen. Record the time base setting (top middle of display):

   ____________ ms/Div

2. Note two consecutive places where the rising waveform is crossing the center horizontal graticule line (these are "positive-slope zero crossings"); the time between these two crossings is one period of the waveform. Estimate the distance, in divisions (1 tenth precision), between these zero-crossings:

   ____________ Div

3. One period of the waveform is the time between the positive-slope zero crossings. Calculate the period, T, in seconds, by multiplying the time-base by the period in divisions.

   T= ____________ ms

4. Compare the value you read with the value reported by the automatic measurements (Add a Time - **Period** measurement on Channel 1).

   T= ____________ ms

5. The frequency is the reciprocal of the period: f = l/T. Calculate the period.

   f = ____________ Hz

6. Compare the value you read with the value reported by the automatic measurements (Add a Time - **Frequency** measurement on Channel 1). (This frequency should be close to the setting on the function generator.)

   f = ____________ Hz

3.5 DC Voltage Measurement Using the Scope

1. Confirm the signal generator output is connected to the Channel 1 input of the scope and confirm the generator's output has an offset of 0 V.

2. Center the generator output on the scope display by pressing the horizontal position knob until you see **0.0s** as the horizontal offset. Then press the vertical position knob until you see the vertical position displays as **Ch (1) = 0.0 V**.

3. Set the function generator **Frequency** to **1 kHz** and **Amplitude** to **1000 mVpp**.

4. Set the generator Offset to **+500 mV DC**.

5. The waveform should be a sine wave raised up from the center of the vertical scale; it is **DC offset** from the zero level of the display. **Estimate the DC offset** (the distance from the display zero level to the middle level of the sine wave) **by using the divisions on the scope**:

   ____________ mVDC

   Estimate the amplitude (peak) of the signal's DC component:

   ____________ mVp

6. Set the signal generator offset as high as it will go and record the offset:

   ____________ VDC

   Set the function generator offset as negative as it will go and record the offset:

   ____________ VDC

7. Now use the large scroll knob to vary the **DC offset** by while you watch the oscilloscope; play with it a bit to see what happens as the DC offset changes.

Another way of measuring DC offset is by using **cursors**. Enable the cursors by pressing the [**Cursors**] button. The cursor menu provides the user with the option to select the tracking mode and channel that each cursor is measuring.

1. Select cursor mode to be "**Manual**". To measure offset, set the offset of the function generator to **0V** and then move one of the cursors so it touches the wave on the very bottom.
2. Set the offset of the function generator back to either **+500 mV** or **–500 mV** and then move the **other** cursor so it touches the wave on the very bottom.
3. The offset is given on the bottom of the screen and is shown under ΔY**(1)** This displays the voltage difference between the first and the second cursors.

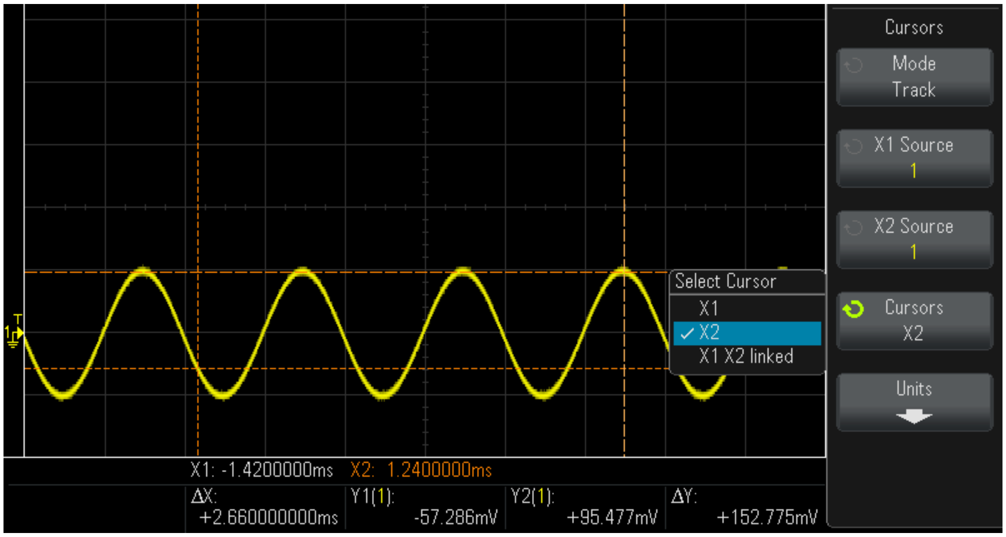


Figure 23: An example of using cursors to measure a sine wave's DC offset

3.6 AC Voltage Measurement with the Digital Multimeter (DMM)

Figure 24

The RMS value of a waveform is the square root of the mean value of the squared function of the instantenous values of the waveform. If a sinusoid has a peak amplitude of A Vp (which is 2A Vp-p), it has an rms amplitude of 0.7071A Vrms. The RMS of a cosine waveform can be calculated by the following relationship:

$$v(t) = A\cos(\omega_0 t) \tag{1}$$

$$
\begin{align}
v_{rms} &= \sqrt{\frac{1}{T}\int_0^T A^2\cos^2(\omega_0 t)dt} \tag{2}\\
&= \sqrt{\frac{1}{T}\int_0^T A^2\frac{1+\cos(2\omega_0 t)}{2}dt} \tag{3}\\
&= \sqrt{\frac{1}{T}\int_0^T \frac{A^2}{2}dt} \tag{4}\\
&= \frac{A}{\sqrt{2}} \tag{5}\\
& \tag{6}
\end{align}
$$

Note: this is a relation for sinusoids only; the ratio of peak to rms is different for different wave shapes.

1. With the signal generator output still connected to the Channel 1 input of the oscilloscope (use a BNC-BNC cable), set the signal generator to a **1 kHz** sine wave with **0 V** offset.
2. Set the amplitude as high as the signal generator will allow.
3. Measure the (peak) amplitude of the sine wave with the scope:

____________ Vp

4. Disconnect the function generator output from the scope and connect the generator to the input of the DMM using a BNC-to-grabber cable to clip into the red and black DMM probes. (**red to red, black to black – be careful to connect to the red DMM input that can measure Voltage – look for a 'V' and the black DMM input to the 'LO' input right below that**).

5. Set the DMM to measure AC volts by pressing [ACV]. In **Range: Auto** mode, the DMM will choose the right range to measure the sine wave. Record the range it chooses.

   ____________ V

6. The DMM shows the units as VAC (Volts AC), this is actually Volts RMS. Record the displayed (RMS) voltage:

   ____________ Vrms

7. Recalling that the RMS amplitude is 0.7071 times the peak amplitude, convert the rms voltage just measured to peak voltage:

   ____________ Vp

   If this value is not the same as in (3), then there is some error. Estimate the error using the peak voltage derived from the DMM reading as the "assumed correct" value:

$$\%Error = \frac{(V_{p\_from\_DMM\_reading} - V_{Oscilloscope\_reading})}{V_{p\_from\_DMM\_reading}} * 100\%$$

   ____________ % Error

3.7 DC Voltage Measurement with the Voltmeter

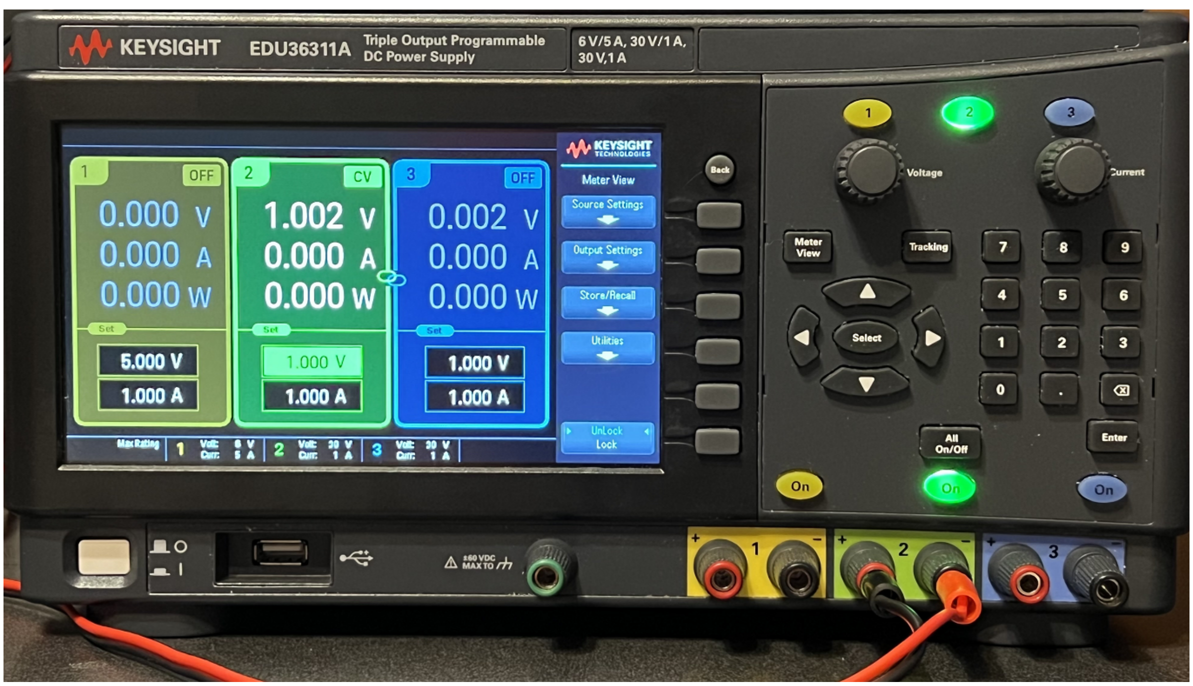
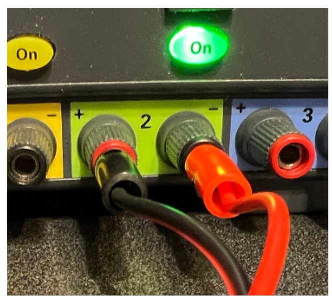


Figure 25: Triple Output DC Power Supply

General Info: A DC power supply that is described as being a "**30 V, 1A**" supply may, in fact, go a bit higher than +30 V and may go a little bit negative; a similar situation exists for a "–30 V" supply, where you connect the positive output to GND (ground) and use the negative output to supply –30V.

Set the DMM to measure **[DCV]** (DC volts), not ACV (AC volts). For each measurement, set the DMM scale to the range that gives you the most significant figures.



1. Connect channel **2** of the Power Supply to **the same black and red inputs to the DMM** (used in the previous section) with two banana-plug-to-banana-plug cables. (See the pictures to the right)

2. Set the output of channel 2 by selecting **[2]** above the Voltage and Current knobs, then set the output voltage to **30V** by typing "30 [Enter]" on the keypad. Enable the output of the Power Supply by pressing the green **[On]** button above the channel 2 outputs.

3. Set the channel **2** output voltage to its maximum. You can do this by typing a number larger than 30 and the Power Supply will set the voltage as high as it is capable of.

4. Record the nominal voltage displayed on the Power Supply:

   ____________ V

   And the voltage measured by the DMM:

   ____________ VDC

5. Set the channel **2** output voltage to its minimum, **0V**. The voltage will not be exactly 0V. Record the nominal voltage displayed on the Power Supply:

   ____________ mV

   And the voltage measured by the DMM:

   ____________ mVDC

6. Swap the red connector and black connector on the channel **2** outputs of the Power Supply, red cable into black (negative) output and black cable into red (positive) output.

7. Set the channel **2** output voltage to its maximum, measure the voltage (negative):

   ____________ VDC

8. Set the channel **2** output voltage to its minimum, measure the voltage (negative, close to zero):

   ____________ mVDC

3.8 Breadboard DMM Measurements

In this class, we will be using digital multimeter and the oscilloscope as the two main measurement instruments. These devices are used to measure different kinds of electrical properties and patterns. The power supply provides DC voltages between -30V and +30V at currents as high as 5A. The DMM is a tool we use to measure the voltage across a component in circuit. The DMM can measure DC voltages as well as root-mean-square (RMS) values of AC voltages. To measure a potential difference, the DMM positive and negative probes are connected in parallel with the device under measure. The oscilloscope plots voltage over time and can be used to measure many different properties of periodic (oscillating) signals.

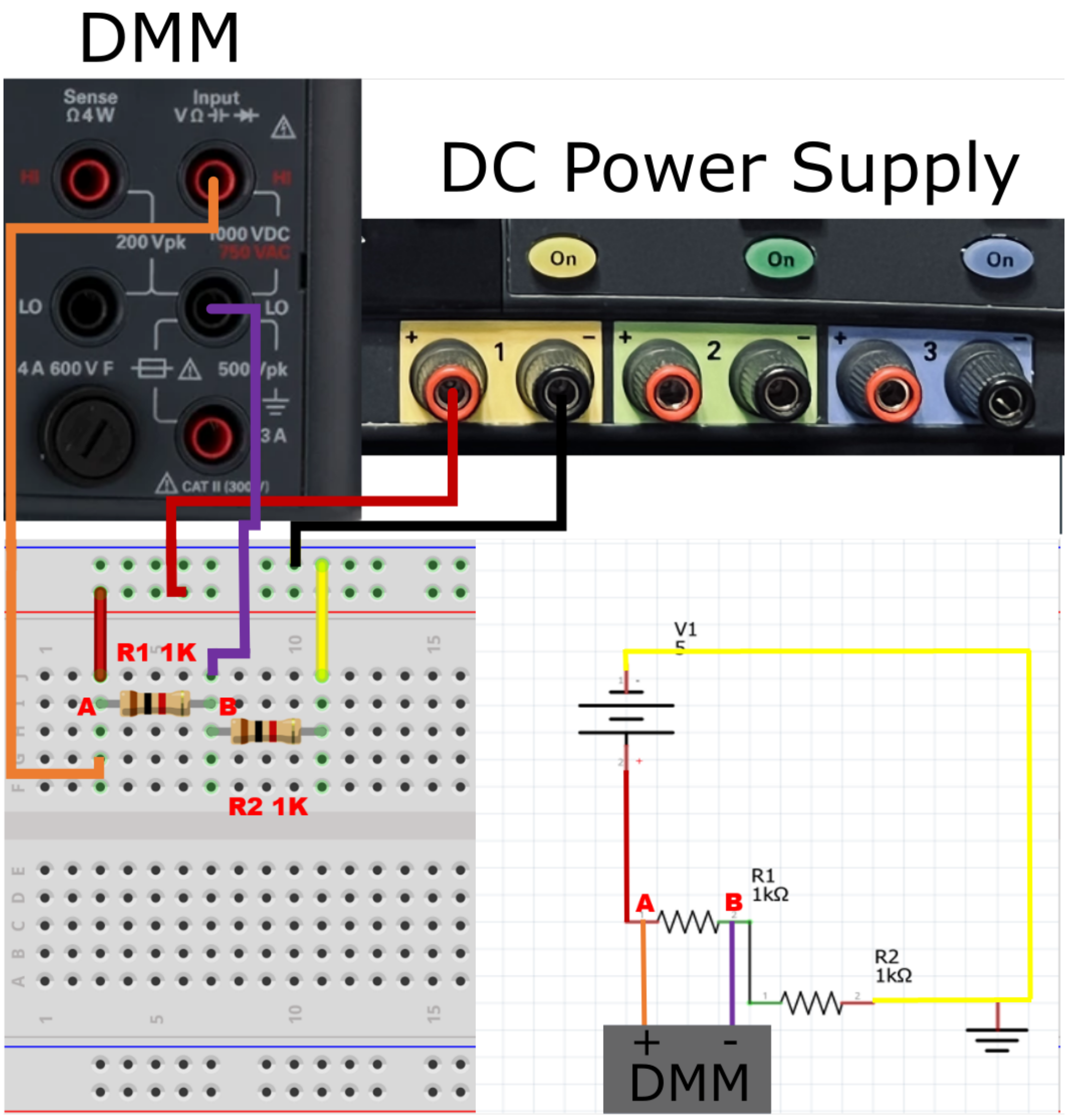


Figure 26: An example of circuit connection with schematic

Breadboard and measurement introductory exercise:

1. Assemble the circuit in Figure 26 on a breadboard, using the schematic to guide you on how each component should be connected to the circuit.

In Figure 26, a 5V DC voltage from the DC power supply is placed across two equal resistors, R1 and R2 in series. The multimeter probes (port Input HI and Input LO) from the DMM is connected in parallel to resistor R1 (DMM and R1 share nodes A and B) to measure the voltage drop across R1.

For how to interpret resistor color codes, see Section 3.9.

2. Connect to the circuit to +5V from channel 1 of the DC power supply, the top red wire in Figure 26.
3. Connect the ground of the circuit to negative terminal from channel 1 of the DC power supply, the black wire in Figure 26.
4. Enable the channel 1 output of the power supply, then use the DMM to measure the voltage across each resistor:

	Voltage(v)
R_1	
R_2	

Table 2: 2-Resistor Circuit Results

5. Ask the Teaching Assistants to verify your values.

3.9 Tips for Reading Resistor Codes

www.resistorguide.com

Color	Signficant figures			Multiply	Tolerance (%)	Temp. Coeff. (ppm/K)	Fail Rate (%)
black	0	0	0	x 1		250 (U)	
brown	1	1	1	x 10	1 (F)	100 (S)	1
red	2	2	2	x 100	2 (G)	50 (R)	0.1
orange	3	3	3	x 1K		15 (P)	0.01
yellow	4	4	4	x 10K		25 (Q)	0.001
green	5	5	5	x 100K	0.5 (D)	20 (Z)	
blue	6	6	6	x 1M	0.25 (C)	10 (Z)	
violet	7	7	7	x 10M	0.1 (B)	5 (M)	
grey	8	8	8	x 100M	0.05 (A)	1(K)	
white	9	9	9	x 1G			
gold			3th digit only for 5 and 6 bands	x 0.1	5 (J)		
silver				x 0.01	10 (K)		
none					20 (M)		

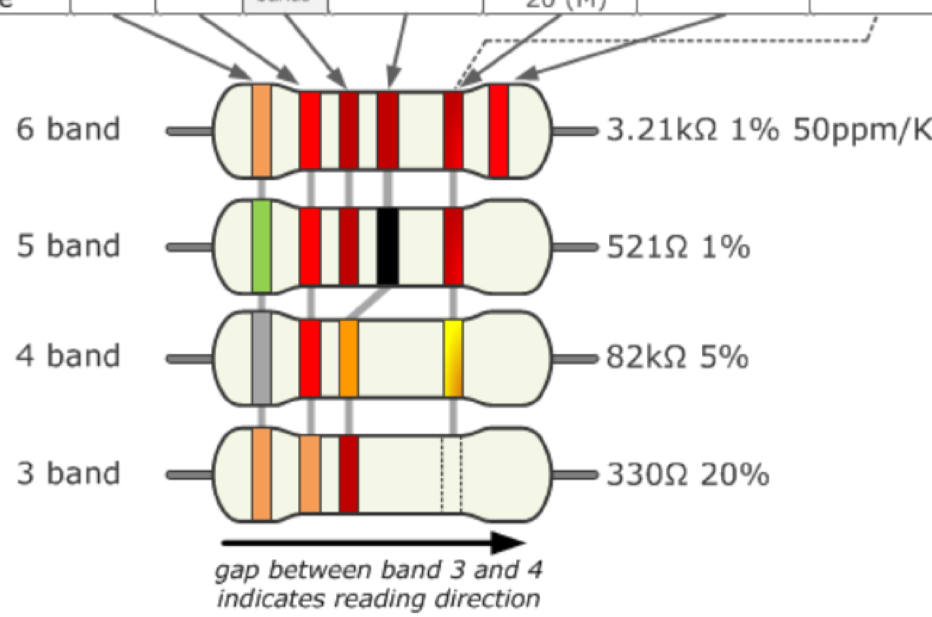


Figure 27: An example of circuit connection with schematic

Here are some general tips for reading the color code:

The reading direction might not always be clear. Sometimes the increased space between bands 3 and 4 provide an indication of the reading direction. Also, the first band is usually the closest to a lead. A gold or silver band (the tolerance) is always the last band.

It is a good practice to check the manufacturer's documentation to be sure about the color coding system used. When in doubt, measure the resistance with a ohmmeter. In some cases this might even be the only way to figure out the resistance; for example when the color bands are burnt off.

3.10 Resistive Networks

Parts List		
Part	Value	Quantity
Resistor	10Ω	1
Resistor	10Ω	1
Resistor	1kΩ	1
Resistor	1kΩ	1
Resistor	34Ω	1

1. Assemble the circuits in Figure 7 on a breadboard. When the circuit is assembled it should look like Figure 28.

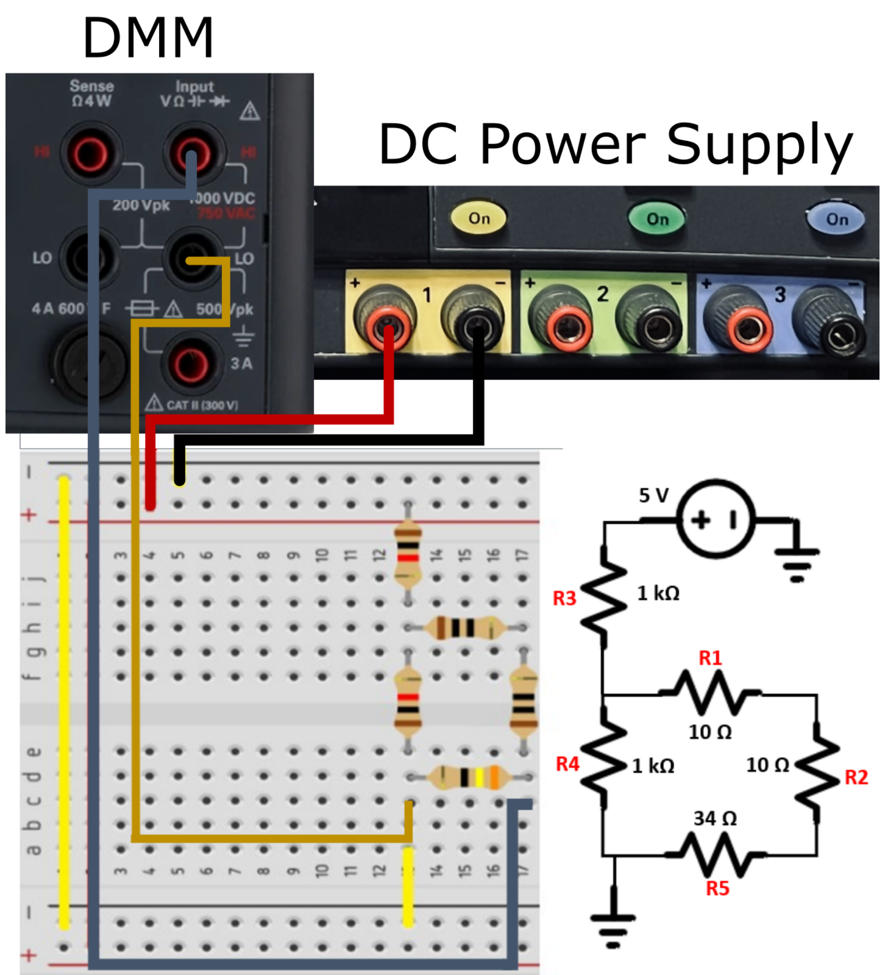


Figure 28: An example of circuit connection on breadboard measuring voltage across R2

For how to interpret resistor color codes, see Section 3.9.

2. Connect to the circuit to +5V from channel 1 of the DC power supply, the red wire in Figure 28.

3. Connect the ground of the circuit to negative terminal from channel 1 of the DC power supply, the black wire in Figure 28.

4. Use the DMM to measure the voltage across each resistor:

	Voltage(v)
R_1	
R_2	
R_3	
R_4	
R_5	

Table 3

5. Ask the Teaching Assistants to verify your values.