Santa Clara University	ELEN 164	Dr. Shoba Krishnan Ben Lampe
	Laboratory Equipment	

I. Objectives

- To learn the common lab equipment for measuring and characterizing components.
- To become comfortable operating oscilloscopes and function generators, and understanding their settings and quirks.

II. Laboratory Procedure

Digital Oscilloscope (Scope) (Agilent – DSO-X 3014A)
This is the device that you will use to learn about more interesting circuits—ones that have voltages changing in time, and ones that require you to look at multiple voltages at once .

The oscilloscope has four channels to measure voltages with four different probes at the same time. **There should be two oscilloscope probes at each station (ask your TA if it not the case)**. Additionally, there should be one BNC cable.

When you want to measure an external voltage signal, you must connect the ground clip to a suitable place in the circuit (almost always the ground node) and then connect the input probe to the point you want to investigate. **When you use two probes at the same time, make sure ground clips are connected at the same point. The ground clips of the oscilloscope are connected through the chassis to earth ground, so if you have ground clips on multiple nodes, THEY WILL BE SHORTED TOGETHER THROUGH THE SCOPE (BAD!!!). Power supplies do the same thing, so connect your scope ground to the same ground as the power supply. For this reason, scopes cannot make differential measurements like multimeters can.**

Setting up a scope

Let's turn on the oscilloscope! On the screen you should see two axes. **The y-axis shows voltage and the x-axis shows time**. This screen shows how the voltage across your probe changes in time. With nothing connected, your scope is just measuring noise, which is what appears on the screen. Press the "1" button to make sure channel "1" is on and to bring up the menu for the channel. Test that pressing it again will turn that channel off. Pressing the "2" button will turn on channel 2 and bring up the menu for channel 2. Channel 1 appears in yellow, channel 2 in green, and each other channel has its own color. Set up the scope so only channel 1 is on, and the channel 1 menu is shown on the bottom of the display.

Let's go through the options on the menu. They can be adjusted by pressing the corresponding soft key under the display, and adjusted by pressing it more times or using the "push to select" dial on the right center of the display.

First is the "**Coupling**," which can be AC or DC. DC is the default, and connects the scope directly to the signal it is sampling. AC coupling is the equivalent of connecting to the signal through a large capacitor: the DC value is filtered out, so you only look at the AC changes in the signal. This is helpful for looking at noise or ripple in a large DC signal. **Set the scope to DC coupling**.

The next option is the input **impedance** of the scope. This can be either 1MOhm or 50 Ohm. 50 Ohm is used for high frequency or high fidelity measurements, especially in RF, where everything in the system needs to be matched to 50 Ohms to prevent reflections. 1MOhm allows for a wider input voltage range. **Make sure to use 1MOhm**, which can handle 300Vrms (50Ohm impedance can only handle 5Vrms).

The next 3 settings are used to limit the bandwidth, switching between fine and coarse adjustments for the X and Y scales (easier to do by pressing the associated knobs), and inverting the signal. These can all be off.

The last button opens the menu for the **probe**. You can change the channels units (i.e. if you are using a current probe), and scale the probe's output if you are using a high impedance probe (which includes a 10:1 attenuator, increasing the range of voltages you can measure). **Keep the units in volts and the probe as 1:1**, and ignore the skew and probe check settings. You can press the back button in the bottom left to return to the channel menu. Each of the 4 channels has its own menu where these settings can be customized.

Now that the input is set up how we like it, we need to adjust the display for the signal we expect to see.

Notice that the top of the display shows the vertical scale (volts per division) for each channel, followed by the horizontal offset and scale (seconds per division). The last few bits represent the trigger settings. The first signal we will measure is coming from the Demo 2 port on the oscilloscope, and it is a 1kHz square wave with an amplitude around 2 volts. Turn the vertical SCALE knob (just above the channel selection button) and observe that you can change the voltage scale on the display. Adjust it to 1V per division. The 1kHz square wave has a period of 1ms, so adjust the horizontal knob, at the top left of the control panel, so that the scale is 500us per division (a period is stretched across 2 divisions). Now for the trigger…

Trigger

The trigger is one of the least well-understood parts of the scope for new users, so it gets its own section. A scope can only usefully display periodic data: waves and signals that repeat themselves. If the signal is not periodic, the display will be constantly changing, will be indistinguishable from noise, and won't be very useful. This is because the entire data capture mechanism is based around a "trigger" event. The scope saves data into a buffer the same width as the display. When the trigger is activated, the latest buffer of data is flushed to the screen. If the signal is periodic, and activates the trigger once per cycle, then the data flushed to the screen will look the same every time, giving the appearance of a stationary, static wave that can be interpreted and analyzed.

Locate the "Trigger" section of the control panel, and press the "Trigger" button to open the trigger menu. The first option is the Trigger Type, which defines the kind of event that activates the trigger.

There are many, and they get complex, but the basic one that we will use (and the default) is Edge Triggering. With this method, you set a voltage level ("trigger level"), and when the signal passes that level, the trigger activates.

The next setting is which channel the scope should use to trigger events. Keep this on channel 1. The final setting is the slope, which can be a rising edge, falling edge, alternating, or either. Rising edge means the trigger event happens each time the input signal rises above the trigger level. Falling edge would be when it falls below trigger level. Alternating and either are exactly what they say they are (trigger on alternating, or either rising or falling edges).

Common trigger issues involve setting it too high, where the input signal never triggers the scope, or setting it too low, where random bits of noise will trigger it, causing your signal to jump across the screen randomly. Another issue is if you have a signal that rings, and can cross the trigger level multiple times per cycle. This will cause the signal to jitter back and forth slightly. Keep positive edge trigger on channel 1. For the Demo 2 signal, which goes from 0 to 2V, set to trigger to the middle of the rising edge (so around 1V) using the trigger dial.

Now that everything is set up, you're ready to start measuring! Always follow these steps when you turn on your scope, before you start trying to measure anything:

1. Set up the appropriate channel settings, like impedance and coupling.
2. Determine the scale on which you need to be measuring.
3. Figure out how the trigger needs to be set

After performing all of the above setup, connect CH1 of the scope to the probe calibration port (Demo 2) and observe the resulting waveform.

Measurements

There are two measurement options that you can choose. Quick measurement allows you to measure commonly measured parameters such as amplitude, RMS value, and frequency. Pressing the "Meas" button opens this menu, where you can select the measurement to perform and the channel to perform it on.

There will be times when you have to use cursors to measure. Press the "Cursors" button to access the cursor menu. Keep them on manual mode, but you can select which channel to use them on, and whether you want a cursor that can be scrolled along the X axis, or one that can be scrolled along the Y axis. you need to use select button to select each cursor to move it to the location you want. The "cursors" box on the screen provides **Δtime** and **Δvoltage**. From those two values, everything can be measured similar to quick measurement options. **Record volt/div and time/div EVERYTIME you take capture of the screen shot**. Without these two values, the screen capture means nothing.

Use cursors to measure the peak-to peak amplitude of the Demo 2 signal. Use the built-in measurement function to find the frequency. Take a picture or screenshot of the scope display

Math
The Math button opens the math menu, which allows you to perform operations on one or multiple channels, such as adding or multiplying channels together, or taking FFTs to find frequency components in a signal you are measuring. The settings allow you to select the operator, the source channel(s), and any other settings for the math function you are performing.

Take an FFT of the square wave on your screen. Set the span (the range of frequencies you are looking at) to 10kHz, and the center frequency to 5kHz. This means the center division corresponds to 5kHz, and the frequency stretches for 5kHz to either side (so from DC-10kHz, with each div representing 1kHz).

The vertical scale and offset can be adjusted with the dials next to the button for the math menu. For the FFT, the vertical scale is in dBV, or decibels relative to 1V. This is calculated by taking 20 times the base-10 logarithm of the RMS value of the input signal, dBV = 20log(Vin_rms).

What is the frequency and magnitude of the strongest frequency in the input square wave signal? What do you notice about the frequencies that make up the square wave?

Section 4: Function Generator (Agilent - 33220A)
The scope is used to observe signals within a circuit. Now we will look at a tool used to generate inputs. The function generator can create various shapes from sinusoidal waves, to square waves, to sawtooth waves and so on. This tool is usually used to send known input to your device and measure the response from this known input.

Turn on the function generator and do the following.

- Press the **SINE** wave button to generate sine wave.
- Find the **AMPLITUDE** button and press it. Turn the knob or use the buttons to set the magnitude to 1.4Vpp. Use the **FREQUENCY** setting to set the frequency to 1kHz.
- Find and press the DC **OFFSET** button and make sure is set to 0V, if not, turn the knob and set it to 0V.
- Do not press the **OUTPUT** button yet! It should not be illuminated. Do not apply an signal until everything else is set up, as having an active signal while you are still connecting your test setup can cause voltage/current spikes or short circuits.

Now you are ready to examine the signal in the oscilloscope.

- Use a BNC cable to connect the OUTPUT terminal on the function generator to Channel 1 on the scope. The scope settings used to view the square wave should work just as well here.
- Press the OUTPUT button to enable the output from the FGEN.

OH NO THERE'S AN ISSUE!!

- You set the FGEN output to 1.4Vpp, right??
- What's the actual peak-to-peak voltage of the signal appearing on the scope?

This comes back to the 50 Ohm vs. 1MOhm termination issue we discussed with the scope inputs. FGENs are commonly used in high-frequency circuits, where everything is matched to 50-ohms: sources have 50 Ohm source impedance, lines have a 50 Ohm characteristic impedance, and loads are 50Ohms. So, our FGEN decides to include a 50 Ohm source impedance, and it expects to drive a 50 Ohm load, and the 2 form a voltage divider. Therefore, to create a 1.4Vpp signal at the load, it actually applies 2.4Vpp to it's source impedance, expecting it to be divided in half. By forcing it to drive a large 1MOhm load, the voltage divider is unbalanced in favor of our large load, and we see the full 2.4Vpp signal.

We can fix this, though! Press the "Utility" button on the scope, and press the soft button that selects "Output Setup." From here the leftmost soft button can be pressed to change from a 50 Ohm load to a "High Z" load. Then you can go and re-set the desired output voltage to 1.4Vpp, and the FGEN will behave properly. NOTE: Every time you shut off the FGEN, it will default back to the 50 Ohm termination mode, and you will either have to change this setting, or dial in half of the Vpp you actually want.