# OPTIX Module 1 – Basic

## Introduction to Common Lab Equipment

Dane Tippett, Michaela Kleinert

## 1 Objectives:

In this module you will learn how to use common lab equipment like

- power supplies;
- function generators;
- oscilloscopes; and
- multimeters.

This module is part of our Modern Physics curriculum and prepares you for the more advanced modules that you will encounter this semester in Modern and later on in ATEP. It is meant to give you a certain familiarity with equipment that is used frequently in a research lab.

Use this manual as you work through the module to keep track of your notes and thoughts. In addition, you may have to add a few printouts or refer to data tables or additional notes in your lab notebook. I'd encourage you to create a Jupyter Notebook for your calculations and plots. Make sure to add all your printouts to the folder in which you keep this manual. Lastly, note that this lab has no formal lab report. Instead, you will turn in and be graded on your notes in this manual.

## 2 Tests and assessment:

In preparation for this module, read through the whole manual and answer the questions that are marked with a *. When you come to lab, be prepared to discuss your answers to these questions with your classmates and your instructor. You should also watch the `VIDEOs` that are posted on our website (`www.willamette.edu/cla/physics/info/NSF-OPTIX`. They are meant to accompany this manual and will show you some critical steps of the module.

In addition, in order to assess the success of this module, you will take a short assessment test before you start ("pre-assessment"), and another one after you have successfully completed this module ("post-assessment"). At this point you will also have the opportunity to provide us with feedback about the module that we will use to improve it for the next student generation. Thank you for your support!

## 3 Equipment:

For this module, you will need

- two power supplies;
- a multimeter;
- a function generator;
- an oscilloscope;
- a breadboard, an LED, and a 400 Ω resistor (or similar);
- various wires (alligator clips, banana cables, BNC cables).

The lab equipment can be found in the OPTIX lab, along with the necessary power cords, alligator clips, and BNC cables to operate these instruments. **Please handle the equipment carefully** as it can easily break or be destroyed if you drop it. Always make sure that all cables are unplugged before you move the equipment, and set things down gently on the table. You should also **ground the optics table** as static charge from the surface can potentially damage the electronics in these instruments and destroy them. Finally, read the instructions in this module *carefully* as it can be very easy to turn a knob too far and burn out electronic components.

# 4 Power Supplies and Multimeters:

Power supplies and multimeters are fairly simple but important pieces of equipment that you will encounter quite often in a research lab. In fact, you probably have some experience with both from your time in the Intro Physics II labs. The easiest way to learn about these two instruments is to use them together, since a multimeter allows you - among other things - to measure the voltage or current a power supply provides. We'll start with just the power supply by itself, and then we'll incorporate the multimeter to measure the output of the power supply in a more systematic way.

## 4.1 Building and characterizing a simple circuit

Many experiments in the lab – and almost any electronic device for that matter – depend on some sort of power supply. Outside the lab environment, one of the most commonly recognizable power supplies is the simple **battery**. Batteries are used everywhere, from powering your car's headlights and radio to keeping your smoke alarms working (assuming you haven't pulled the batteries out because the alarm keeps beeping even though the house isn't on fire!). And even USB cables that you use to charge your phone or AC-to-DC adapters to charge your laptop are just different ways of supplying your electronics with power.

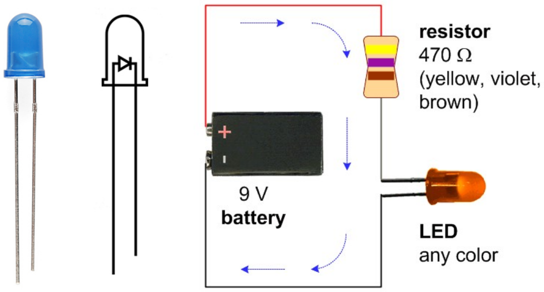
In the lab, you probably recognize a power supply as one of those metal boxes with the big knobs that allow you to adjust the voltage and current, and the small ports that you can use together with banana or alligator cables to bring power to your electronic circuit, see the photo below.

These power supplies are called **Bench(top) Power Supplies**. Most of the OPTIX modules you will encounter this semester or in ATEP will use a power supply because power supplies grant greater control over the voltage and current applied to a device than a battery. Locate the power supply in the OPTIX lab (it should be in the same cupboard as the blue box labeled 'Module 2', on the second shelf) and plug it into an outlet.

Since you will be working with voltage and current, give yourself a quick refresher on Ohm's Law and on how current and voltage are related to the power in a circuit.

*** Ohm's Law and electric power in a circuit:**

To see how a power supply works and what happens when you adjust the voltage and current, let's construct a small circuit. Take a breadboard and construct the circuit shown on the right of the figure below (taken from `www.electroschematics.com/2573/led-circuit/`) by putting a resistor in series with an LED and insert power and grounding wires into the breadboard. Instead of the battery shown in the figure you will use the power supply.



Take two alligator clip cables from the drawers (these cables have the jagged clips that resemble an alligator's mouth) and connect them to your power supply. It is conventional to color-code the cables you use (because that makes trouble-shooting circuits easier): Red cables are connected to the more *positive* voltage side of the circuit while black cables are connected to the more *negative* one. Very often, this more negative side is also connected to ground.

The **LED** (Light-Emitting Diode) is a directional device, which means it will only work properly if the more positive voltage is connected to the correct *lead* of the LED. This lead is typically a bit longer than the other one (unless someone clipped it off!) as you can see in the image above on the left. But even if the leads of the LED are no longer pristine, the inside of the LED also tells you which lead needs to be connected to the more positive voltage: it is the one that's connected to the smaller metal part inside the LED.

Now connect the power supply to the simple LED circuit using two alligator clip cables. Before turning on the power supply, make sure to **rotate all of the knobs all the way counter-clockwise** to ensure that no current will flow, even when the main power button is pressed. Then turn on the power supply by pushing the power button.

**Does the LED light up? If not, why not?**

Now rotate *only* the current knob clockwise (but don't touch the voltage knob).

**Does the LED light up? If not, why not?**

Rotate the current knob all the way counter-clockwise again and then rotate *only* the voltage knob clockwise.

**Does the LED light up? If not, why not?**

**Adjust both knobs until the LED glows. Thus, which physical quantity (current, voltage, or something else) determines whether or not the LED lights up?**

You can adjust this quantity in two ways:

- **Constant Current Mode:** Make sure both knobs are rotated fully counter-clockwise. Then rotate the *voltage* knob *all the way clockwise* and slowly rotate the *current* knob clockwise until the LED starts lighting up.
- **Constant Voltage Mode:** Make sure both knobs are rotated fully counter-clockwise. Then rotate the *current* knob *all the way clockwise* and slowly rotate the *voltage* knob clockwise until the LED starts lighting up.

So, what is the difference between these two configurations, and when should you use which of the two? The answer is in the name: Constant *voltage* vs. constant *current* mode. In **constant current mode**, the power supply will provide as much voltage as is necessary to maintain a constant current. You know this because by rotating the voltage knob all the way clockwise you allow the maximum voltage that the power supply can supply to be available. The current it supplies is then given by the value you dial in with the current knob (that's why you rotate it slowly and observe the LED as you do so). Similarly, in **constant voltage mode**, the power supply will provide as much current as necessary to maintain the voltage you dial in with the voltage knob. It is easy to gloss over this paragraph since the wording is similar, so quickly check with your group members and make sure you understand this difference.

You may wonder why you can't simply dial in some current and some voltage value, but rather, why you should *always* set one of the two dials to its maximum value to maintain a constant voltage or current. Ohm's Law provides the answer because it tells us that the current $I$ and the voltage $\Delta V$ are related through the resistance $R$ of the circuit, $\Delta V = IR$. But in a real circuit, as current flows through it, some of the electrical power is dissipated as heat, which means the temperature of the circuit changes.

**What do you think happens to the resistance in the circuit as the temperature changes? And how will that affect the current and/or voltage in the circuit?**

Using your answer from the previous box together with Ohm's Law, explain the following:

**Why does rotating the current [voltage] knob all the way clockwise ensures that you always have a constant voltage [current] in your circuit?**

Now run the power supply in *constant current* mode. Slowly turn up the current and observe the LED.

**Record the voltage and current readings on the power supply when the LED is well lit.**

$\Delta V_{PS} =$

$I_{PS} =$

**Can you conclude that this is the current flowing through the LED and the voltage across it, respectively? If not, why not?**

## 4.2 Measuring voltage, current, and resistance with a multimeter:

The voltage and current you just recorded in the last box are the total amounts your power supply provides to the *whole* circuit. As you know from Intro Physics II, how much of that voltage or current goes to a single element in a circuit depends on how that element is connected to all the other elements in the circuit. In particular, it depends on whether circuit elements are connected *in parallel* or *in series*.

*** Review Kirchhoff's Loop and Junction Rules:**

In addition, the internal resistance of the circuit caused for example by the wires or the breadboard and the tolerance of your resistor, i.e. the margin of error in a resistor during the manufacturing process, determine the exact voltage across and current through the LED. We can use a **multimeter** to measure these values directly.

Multimeters are often used in the lab to examine and troubleshoot circuits, and to measure voltages or currents before connecting a power supply to another piece of equipment. They are especially helpful when working with complex circuits as every component will affect the overall voltage and current within the circuit. Before you can use the multimeter on your circuit, let's quickly go over its settings (see the photo below for an example, and note that not every multimeter will look exactly the same!).

Starting from the left (at the `OFF` position) and moving to the right, you see two different `V` (= **voltage**) settings, one with a tilde over it ($\tilde{V}$) and one with a straight and dotted line. $\tilde{V}$ is used to measure the voltage of a circuit with *alternating current* (or AC current). In contrast, the straight-and-dotted-line-$V$ symbol measures the voltage of a circuit with *direct current* (or DC current). Most circuits you will be working with in the OPTIX labs are DC circuits, so you will not need to use the AC setting here (but keep in mind that it exists! It's a great way to test for example the voltage coming out of an outlet.). The next setting on the multimeter is the $mV$ setting, which is used to measure small DC voltages (smaller than 1.0 volt) more precisely. Let's measure the voltage of the power supply using your multimeter. To do this, touch the two leads of the multimeter to the ends of the alligator clips (black to black and red to red).

**Record the voltage on the multimeter and explain why there might be a discrepancy between what the power supply says and what the multimeter says.**

Then measure the voltage across the resistor and across the LED by placing the black lead of the multimeter to the more negative side of the resistor/LED and the red lead to the more positive one.

**Record these two values below and explain them (we will revisit these results in a moment).**

$\Delta V_R =$ $\Delta V_{LED} =$

The next setting on the multimeter is for measuring **continuity and resistance**. While you probably know what resistance is (represented by the Ω symbol on the multimeter), you might not know what continuity is (represented by the sideways wifi icon). The continuity function on a multimeter measures how easily current can get from one end of an electronic component to the other. Set the multimeter to the continuity setting by pressing the continuity button (located in the second row just to the right of the yellow button). Turn off the power supply by pushing the general power button (but do not rotate the knobs).

**Take the following components out of the circuit to measure their individual continuities by pressing the two leads of the multimeter against the two ends of each of these components. Observe the multimeter while you measure the continuity, and record any observations together with the reading on the multimeter, which gives you the resistance.**

**(i) one of the alligator cables:**

**(ii) the resistor:**

**(iii) the LED:**

**Explain your observations!**

We will come back to the resistance function in just moment. All the way on the other end of the multimeter are the settings for measuring **current**. One is for measuring larger currents on the order of milliampere or ampere, and the other for measuring very small currents on the order of microampere. But if you simply turn the dial to the current setting, the multimeter will start beeping. This is normal and tells you that the leads are connected incorrectly. To measure current, you have to remove the red cable (plugged in on the very right of the multimeter) and insert it into one of the two ports on the far left. The leftmost port is for measuring larger currents on the order of ampere, while the second leftmost port is for measuring smaller currents on the order of milli- or microampere. Note that the white symbols all have a tilde over them.

**Thus, what kind of current can you measure in this setting? And how can you measure the other type of current?**

Return all components back to your circuit to measure the current flowing through the full circuit and through each of its two elements (resistor and LED). Remember from Intro Physics II that current flows *through* an element in a circuit.

**Thus, what do you have to do to measure the currents? Do this and record your values in the box below. Compare them to the current reading on the power supply ($I_{PS}$) and explain the individual current readings.**

$I_{full} =$ $I_R =$ $I_{LED} =$

Switch back to the **resistance** function on your multimeter.

**Measure the resistance of your resistor two different ways:**

**(i) With the power supply on:** $R_{on} =$

**(ii) with the power supply off (but the resistor still connected to the circuit):** $R_{off} =$

Then take the resistor out of the circuit.

**Measure the resistance of your resistor two different ways:**

**(i) By holding it in both of your hands while touching the multimeter leads to it:** $R_{hands} =$

**(ii) By only touching the leads to it (and not your hands):** $R_{leads} =$

Lastly, calculate the resistance of your resistor in two different ways:

**(i) By By using the current through ($I_R$) and voltage across ($\Delta V_R$) the resistor that you just measured:** $R_{Ohm} =$

**(ii) By using the color coded bands on the resistor (you may have to ask google how to do that!):** $R_{spec} =$

You probably noticed that all of these values are slightly different.

**Compare the individual values. Explain why they are different. What do you think is the most accurate way to measure resistance and what should you do/avoid when measuring resistances?**

**Disassemble the LED circuit and return everything to where you found it when you started this lab.**

### 4.3 How to generate positive *and* negative voltages using two power supplies:

Lastly, you will learn how to connect two power supplies to generate both positive and negative voltages. Quite often, inexperienced students believe that you can extract both a positive *and* a negative voltage from the red and the black output port of a *single* power supply. That is not true.

**Explain why. Hint: Remember that a voltage is a potential *difference*.**

All of our power supplies are **floating power supplies**. That doesn't mean that they hover in air, of course. Rather, it means that they have no *fixed* voltage reference point, and that only the voltage *difference* between the black and the red output port is fixed (and is equal to what the power supply displays). This difference is fixed and highly constant, but there may be an overall voltage offset that is added to both output ports. If, for example, the power supply displays 5.0 V, then this could mean that the black output port is at 0.0 V and the red port at 5.0 V, or that the black port is at 531.1 V and the red one at 536.1 V, or any other example you can think of that has a voltage difference of 5.0 V. In an analogy, think of the length of a ruler (i.e. the *position difference* between the upper and the lower end of the ruler when you hold it vertically). This position difference is constant and independent of the exact location of the ruler: It doesn't matter whether you hold the ruler directly above the floor, at eye height, or in an airplane; the length of the ruler is always the same. Similarly, the potential difference between the two output ports of the power supply is what you dial in, but it is possible that both of the ports have the same additional offset voltage added to them. This feature is actually quite useful if you think about it the other way around: Because the potential difference is fixed and independent of the overall offset voltage, we can *pull* this overall offset voltage to any value we like! Very often, it is pulled to ground (that's why you often connect the more negative output port to ground, which then produces a net positive output voltage from the more positive (red) port). But we can pull it to other values as well as you will see in a moment.

Take a second power supply from the cabinet, rotate all the knobs to the counter-clockwise position, and turn it on by pushing the power button. Prepare your first power supply in the same way. Then set up both power supplies such that they provide a constant voltage of 5.0 V.

**Measure these two voltages with your multimeter and record them below.**

$\Delta V_{PS,1} =$ $\Delta V_{PS,2} =$

Then place the black lead of the multimeter into the more negative (black) output port of your *first* power supply and the red lead of the multimeter into the more positive (red) port of the *second* power supply.

**Record the reading on the multimeter and explain why it is different from the previous two readings.**

$\Delta V_{cross,1} =$

Now take a banana cable and connect the two negative output ports of your two power supplies. Again, place the black lead of the multimeter into the more negative port of one power supply and the red one into the more positive port of the second power supply.

**Record the voltage you measure and explain why the added cable makes a difference.**

$\Delta V_{cross,2} =$

Using this configuration, both power supplies share the same ground and provide you with - in this case - the same positive voltage. Of course, you can now adjust the individual voltages so that e.g. power supply one provides +4.8 V and power supply two provides +3.1 V.

**Quickly confirm this experimentally. Did it work? If not, what went wrong?**

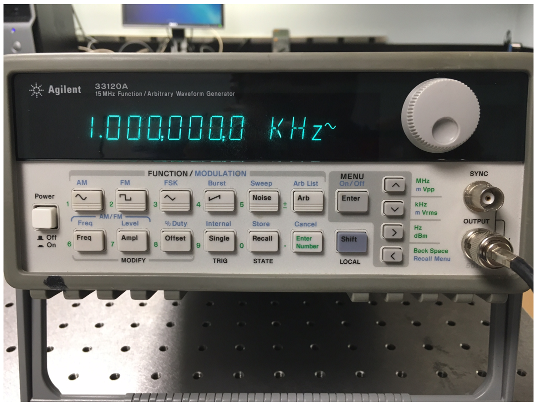
**But how can we generate a positive and a *negative* voltage? Discuss with your group members how you could modify the existing setup to produce +4.8 V from the first power supply and -3.1 V from the second one. Sketch the setup below and then confirm experimentally that it is working as intended.**

# 5 Function Generators and Oscilloscopes:

Function generators, as the name implies, generate periodic electrical signals, for example a sine function, step-function, sawtooth function, or triangular shaped function. To visualize these signals, the output of the function generator is connected to an oscilloscope. Oscilloscopes graph the electrical signal over time and allow you to easily compare multiple different electrical signals by displaying up to four traces at once. In this chapter you will learn about function generators and oscilloscopes, and how to use them together to display and manipulate electrical signals.

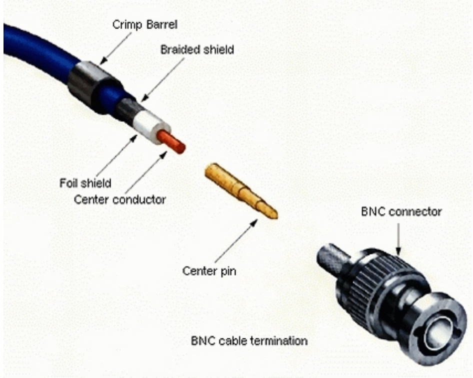
## 5.1 Generating periodic waves with a function generator:

Not all function generators have the same interface, of course, but they all share the same basic functionality, and you should be able to apply what you learn in this module to any function generator you use. The function generator we will use as an example is shown in the figure below (most other function generators have simpler interfaces).



Function generators are devices that generate periodic functions (typically sine, step-function, sawtooth, or triangular shaped) with frequencies that span the range from sub-hertz to several megahertz. A sine function works great to power mechanical wave drivers or motors while a step function is commonly used to start a series of events by providing a *trigger signal*. Quite often, the step function then alternates between 0 V (`LOW`) and 5 V (`HIGH`), which is called a **TTL signal**. Sawtooth or triangular functions provide a linear scan, for example to change the frequency of a laser.

To easily see the periodic signals that are created by the function generator, you will use an oscilloscope. You will learn more about the specifics of the oscilloscope later in this module. For now, its default settings are perfectly fine. Plug the function generator and the oscilloscope into outlets with their respective power cords. Connect the function generator to the oscilloscope using a **BNC cable** by pushing the cable into one of the channels of the oscilloscope and rotating the outer ring until it snaps into place. Repeat with the other end of the cable and the function generator. BNC cables are designed to carry electrical signals over large distances without adding extra noise to them by shielding the signal cable with a grounded mesh, like a Faraday cage. The signal cable is the (golden-colored) center pin that you see when you look into the BNC connector. The silver-colored outer ring is connected to ground once the BNC cable is attached safely to the oscilloscope and function generator. Underneath the black cover of the cable, this ground is then connected to the wire mesh that surrounds a layer of insulation, which in turn surrounds the signal cable (see the photo below, taken from `www.solidfonts.com/bnc-wiring/`).



Turn on the oscilloscope and wait for the startup procedure to complete. Once you see a graph on the screen, press the `DEFAULT SETUP` button located near the top of the oscilloscope interface. This setting is perfectly fine for the following exercises, and you won't need to adjust any of the scaling on the oscilloscope for now (we'll do that later).

**Turn on the function generator.** This particular model will go to its default settings, which is a sine wave with a frequency of 1 kHz (it will be displayed as 1.0000000 kHz on the function generator, and the symbol next to this number tells you that it is generating a sine wave). Note that for other function generators the default setting may be different, though. Confirm that you indeed see a sine wave with the expected frequency by observing the signal on the oscilloscope. There are four buttons on the left hand side of the top row that allow you to toggle between a sine, a square, a triangular, and a sawtooth wave. Push each of these buttons and observe the changes on the oscilloscope.

The large gray knob on the right side of the function generator allows you to adjust the frequency. Rotate the knob and observe how this changes the wave on the oscilloscope (repeat this for a few of the different functions). You might also have noticed the arrows below this gray knob. Pushing these arrows allows you to determine which significant digit of the frequency you change when you rotate the knob, which in turn allows you to dial in very specific frequencies.

**Without changing the axes on the oscilloscope, adjust the generated frequency so that you fit a single full period on the screen. Record this frequency value and the times/division:**

$f_{full} =$

$t/DIV =$

**Then confirm mathematically that $t/DIV$ times the number of divisions that are visible is equal to $f_{full}$.**

Underneath the buttons with the wave icons are three additional buttons labeled `FREQ`, `AMPL`, and `OFFSET`. As you might expect, `FREQ` adjusts the frequency (how quickly the signal oscillates up and down). This button is depressed by default. Similarly, `AMPL` adjusts the amplitude (how large the signal is), and `OFFSET` adjusts the offset (by shifting the signal up and down). Switch to the `AMPL` or `OFFSET` settings and adjust the values by rotating the gray knob. Confirm, by observing the signal on the oscilloscope, that they are doing what you expect them to do. You should now have a pretty good understanding of how to use a function generator. If you have any questions, please check with your group members or instructor before moving on.

## 5.2 Using an oscilloscope to view an electronic signal:

Let's start by familiarizing us a bit with the interface of the oscilloscope. Just as with function generators, the interface of an oscilloscope may change from brand to brand, but the overall functionality is the same, and you should be able to easily apply what you'll learn in this module to other brands of oscilloscopes. The photo below shows you the interface of a typical oscilloscope (this one is made by the company 'Tektronix'). The screen is to the left of the interface and is not shown in the image.

Oscilloscopes are devices that display a constant or changing voltage as a function of time. They thus provide more functionality than a simple multimeter that is mainly optimized for measuring and displaying a single value, i.e. a constant voltage. The oscilloscope you will be working with is a *digital* oscilloscope. *Analog* oscilloscopes were commonly used many years ago, but they have been replaced almost completely by the more light-weight and easier to use digital oscilloscopes. Analog oscilloscopes contain an electron gun (that fires a beam of electrons onto a phosphor display screen) and a set of plates (that deflect the beam of electrons depending on the measured voltage). Digital oscilloscopes use more complicated electronics that convert a voltage signal to a pixel value on a digital screen display. Because these additional calculations take some time, digital oscilloscopes tend to be a bit slower than analog oscilloscopes.

The BNC connectors at the bottom of the interface allow you to connect up to two external signals to be displayed on the oscilloscope (labeled as `CH 1` and `CH 2`). Note that there are digital oscilloscopes that allow you to connect more signals. The `EXT TRIG` connector allows you to attach an external trigger *in addition* to the two signals. This trigger signal will not be displayed on the screen; it simply tells the oscilloscope *when* to start displaying the signals that are connected to `CH 1` and `CH 2`. You will learn more about the trigger and how to use it to display a stable signal in the next subsection.

The display shows you the measured voltage on these two channels as a function of time. Each of the squares represents a unit voltage in the vertical and a unit time in the horizontal direction. The display will also tell how large such a unit voltage or unit time is (for the Tektronix oscilloscope, that information is listed near the bottom of the screen). You can change the size of a unit voltage or unit time by rotating one of the three larger knobs near the bottom of the interface labeled `VOLT/DIV` or `SEC/DIV`, respectively. If you are using both channels (`CH 1` and `CH 2`), you can adjust their scales independently so that the vertical axis of one channel can be on one scale while the other channel can be on a different scale. This is great if you want to compare a very small signal to a much larger signal. If you have a very short signal (like a short burst of laser light) or a very fast signal (like a sine wave with high frequency), you want to "zoom in" by setting the `SEC/DIV` to a small value. Similarly, if you have a small signal, you want to set the `VOLTS/DIV` to a small value. Overall, you want to use the available space on the display as best as possible and avoid the useless "white space" that shows nothing of your signal.

For the following two signals, write down which `VOLTS/DIV` and `SEC/DIV` setting you would use to optimally display the signal. Keep in mind that there are ten divisions in the horizontal (seconds) and eight divisions in the vertical (volts) direction.

**`Signal 1:` You want to measure the width of a very short but intense burst from a pulsed laser. The laser pulse has a duration of 100 ns and an intensity, as measured by a photodetector, of 10 V.**

`VOLTS/DIV` =

`SEC/DIV` =

**`Signal 2:` Now you want to experimentally verify that this laser sends out such a burst of light with a *repetition rate* of 10 Hz.**

`VOLTS/DIV` =

`SEC/DIV` =

In addition to changing the scale, you can also shift the signal either up/down or left/right by rotating the smaller knobs above the large knobs labeled as `POSITION`. For example, if you display two signals on the screen, it can be advantageous to adjust the vertical position of one of the signals to separate the two signals and thus making it easier to view both signals simultaneously. Or, it could be advantageous to overlay the two signals to compare them more easily. Similarly, you can shift the signal to the left or right, for example if you want to investigate what happened before or after a certain time. Play with these knobs and make sure you understand what they do before you move on.

### 5.2.1 The trigger:

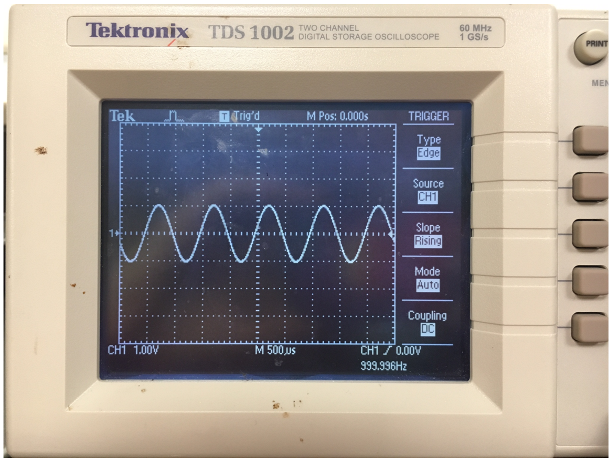
We are now ready to investigate the most important setting on the oscilloscope: the **trigger**. The trigger tells the oscilloscope when to start displaying the signal. By default, it is set to 0 volts. That means that the oscilloscope will trigger and start displaying the voltage signal every time the signal passes through this 0 V threshold. Thus, the screen is refreshed in sync with the frequency of the signal you are displaying. When the signal does not pass the 0 V line, the oscilloscope refreshes at a frequency that is determined by the time duration you are displaying. For example, if your time divisions are set to 1 second, the oscilloscope will refresh the display every 10 seconds because there are 10 divisions, which corresponds to a frequency of 0.1 hertz. Thus, the refresh rate of the oscilloscope and the frequency of your signal are out of sync, and you will see a rapidly changing signal.

**Test this statement by displaying a sine function with a frequency of 1 Hz and shifting it up by increasing the offset. Describe what you see.**

You can think of the trigger as a **strobe light**. Imagine your friend is jumping up and down in a dark room once every second. If you flash on a light once every second, you can make it look as if your friend is hovering in air the whole time, because the rest of the motion happens in the dark. But if you flash the light on at a different frequency, for example once every tenth of a second, you will see your friend in motion (jumping up, falling down, etc.).

The right-most column of the interface allows you to adjust the trigger settings for the oscilloscope. The top knob controls the trigger level, i.e. the voltage value at which the oscilloscope triggers. You can see the trigger level as a small arrow on the right side of the screen. Move the trigger level up and down and you should see the small arrow move up and down as well.

By default, the oscilloscope is set to trigger on a *rising slope* and a trigger level of 0 V. That means that the signal has to pass this trigger level of 0 V on a rising slope to trigger the oscilloscope, i.e. from negative to positive voltages. When the signal passes the 0 V mark from a positive to a negative voltage, the oscilloscope will not trigger. To change the slope on which the oscilloscope triggers, press the `TRIG MENU` button underneath the trigger level adjustment knob. On the screen, you will see five options off to the right side of the oscilloscope graph called `TYPE`, `SOURCE`, `SLOPE`, `MODE`, and `COUPLING`, from top to bottom. Next to these options are five physical buttons, each corresponding to the option directly across from it. The center option is for the `SLOPE` and allows you to toggle between a rising and a falling slope by pressing the corresponding button. The interface for the oscilloscope when in the `TRIG MENU` is shown in the figure below.



Let's play with the **slope** of the trigger signal to fully understand how it works. First, set the oscilloscope to trigger on a *rising* slope. Set the trigger level to 0 V and set the function generator to a sine wave with a frequency of 1 Hz and an amplitude of 1 V. Adjust the offset to 0 V.

**Carefully sketch the wave that you see on the oscilloscope. Especially indicate which part of the wave passes through the origin on the screen.**

Now set the oscilloscope to trigger on a *falling* slope.

**Again, carefully sketch the wave that you see. How does the signal change? Explain these changes.**

**Lastly, while on either rising or falling slope, slowly change the trigger level and observe the changes. Summarize them below.**

The top button in the `TRIG MENU` is labeled `TYPE` and changes the triggering type of the oscilloscope. So far you've been using the `EDGE` setting. Another important setting that allows you to measure short voltage bursts (for example short laser pulses) is the `PULSE` setting, where you can tell the oscilloscope to trigger on signals that are shorter or longer or equal to a certain time duration. I am happy to show you how to use it if you are interested (or feel free to play with it if you have time), but since this is an advanced setting we will skip it in this module.

Let's now explore the `SOURCE` option, which allows you to select the signal that is used as the trigger. If you played with the `TYPE` setting, make sure to switch your oscilloscope back to the `EDGE` trigger type. Toggle through the `SOURCE` options; you should see that you can trigger on either of the two channels as well as on `EXT` (or `EXT/5`) or the `AC LINE`. `EXT` is exactly what it sounds like: If an external signal is applied to the third BNC connector on the bottom-right of the oscilloscope, then the oscilloscope will trigger whenever that signal fulfills the trigger requirements. This is handy if you have a general trigger signal that starts your experiment and you want to measure when, with respect to this general trigger signal, the signals on the two channels appear. `AC LINE` allows you to trigger on the frequency of the AC power line, which in the United States is 60 Hz (Europe and most other countries use 50 Hz). As you change to this setting, your sine wave will move around erratically and be no longer stable or stationary. That makes sense, because chances are the frequency that you have dialed in with the function generator is different from the frequency of the AC power line, so the two are out of sync. Carefully adjust the frequency of your function generator until the sine wave is stable again.

**Record this frequency (it should be close to 60 Hz, of course!):** $f_{AC} =$

**Can you also stably display a sine wave of twice the frequency (i.e. 120 Hz) using the AC trigger? How about of half the frequency (i.e. 30 Hz)? Try it and record your conclusions below.**

Triggering on `AC` is a great way to display a least some starting signal when you don't know yet what the amplitude or frequency of your actual signal is. But it also has a more useful application: Electrical noise at the AC power frequency is often picked up on your signal cables, especially when you use simple banana or alligator clip cables (remember that BNC cables shield a lot of that noise by design!). Alternatively, your detectors can pick up this kind of noise, for example, if you record a weak signal with a photodiode and have the room lights on, you will probably see a sine wave with a frequency of 60 Hz added to your signal, because the room lights oscillate with that frequency. Triggering on the `AC LINE` allows you to easily determine if the noise you see on your signal is really 60 Hz (which means it comes from the AC power line), or if it has some other frequency (and thus comes from somewhere else).

We will now investigate the `MODE` option. Here, you can choose between `AUTO`, which is the default, and `NORMAL`. In `AUTO` mode, the oscilloscope triggers continuously and is forced to trigger even when it does not receive an appropriate trigger signal within a certain amount of time. For many applications, this is what you want to use, because the signal on the display refreshes continuously and you can easily see what kind of signal you are working with. In contrast, in `NORMAL` mode, the oscilloscope waits for the trigger level and displays the signal only once; even if it receives appropriate trigger signals afterward, it will not refresh the display. This option is fantastic if you have a signal that contains multiple valid trigger points, but you are only interested in what happens near one of these trigger points. For example, picture a signal that is at the 5 V level for a while, then switches to 0 V, switches briefly back to 5 V, back to 0 V, and then back to 5 V.

**Draw this trigger signal.**

Such a signal may seem quite arbitrary, but you can probably find it (or a very similar kind of signal) in many research labs. An example could be a laser experiment. Imagine that a laser is on at the beginning and the end of an experiment (signal is at 5 V), but is turned off during parts of the experiment (signal is at 0 V), and is briefly flashed on during the experiment (signal briefly jumps up to 5 V). You might be interested in the time immediately following to the brief 5 V period during the actual experiment, and you would want the display to show that time interval. Choosing a rising slope makes sense (because that forces the oscilloscope to trigger when the signal changes from 0 V to 5 V.

**But what happens if you set your trigger to a rising slope and are in `AUTO` mode? Remember, in that mode the oscilloscope refreshes the screen every time it receives a valid trigger signal. In contrast, what happens when the oscilloscope is in `NORMAL` mode?**

Lastly, you can choose different kinds of `COUPLING`. These different options allow you to filter out noise from your signal and avoid false triggering events. For example, setting the `COUPLING` to `DC`, the default, allows all frequency components of your signal to pass through and be displayed. For many applications this is a good choice. But if your signal has a lot of high-frequency noise, then `HF REJECT` can be advantageous, because it filters out all frequency components above about 80 kHz.

**Confirm this experimentally by displaying a sine wave and selecting `HF REJECT`. *Increase* the frequency of your sine wave from 10 kHz to 20 kHz to 30 kHz and so on. Observe what happens to the wave, especially as you pass the 80 kHz threshold, and record your observations below.**

Similarly, `LF REJECT` blocks all DC signals and attenuates signals with frequencies below 300 kHz.

**Again, display a sine wave, select `LF REFECT` this time, and slowly *decrease* the frequency of your wave. Record your observations below.**

`AC` works similar to `LF REJECT`, but it only blocks DC components and attenuates signals below 10 Hz.

**Again, confirm this experimentally and record your observations below.**

If you'd like to learn more about the trigger and its options, have a look at this fantastic webpage: `www.tek.com/manual-topic/trigger-controls`, which is designed and maintained by Tektronix, the manufacturer of your oscilloscope.

### 5.2.2 AC or DC Coupled Signals:

Adjust your trigger level and settings such that you can display a stable sine function from your function generator. We'll now focus on the `COUPLING` option that you can access when you press the `CH 1 MENU` button (assuming that the signal from the function generator is connected to channel 1). It is the first option from the top. Overall, the coupling options here are similar to the coupling options we just explored for the trigger, but instead of affecting the trigger signal, they affect how the actual signal is displayed. The most important settings are `DC COUPLING` or `AC COUPLING`. `DC COUPLING` means that every signal, independent of its frequency, is displayed on the screen. In contrast, `AC COUPLING` displays only AC voltage signals and suppresses DC components, i.e. (essentially) constant voltages. This is very similar to the `AC` setting in the trigger menu: Every frequency component below 10 Hz is filtered out by a high-pass filter, and that includes of course any DC voltage.

Display a square wave and lower its frequency from about 100 Hz down to the single hertz range. Toggle between `DC` and `AC COUPLING`.

**How does the signal change when you get to low frequencies? Describe the changes and sketch the wave function. Then switch to the other three wave functions and repeat. Again, describe the changes and draw sketches for all remaining wave functions.**

So what is happening here? Again, in `AC COUPLING`, all low frequencies are suppressed or blocked. To understand how suppressing the low frequency components affects your signal, we need to briefly review *Fourier Analysis* and *Fourier Sums*. Remember, the basic idea of Fourier Analysis is that *any* signal can be decomposed into a sum of sines and cosines of appropriately chosen amplitude and frequency. Go to `www.falstad.com/fourier` to see this idea in action. In this applet, you can pick the original wave form (sine, cosine, triangle, sawtooth, or square). The applet will then display the amplitude of the sine and cosine components that contribute to this signal. Select the *square* option. Then adjust the slider called *Number of Terms* to select how many of these frequency components are included. For example, if the number of terms is set to one, only the first cosine term will be displayed (and as you can see, for this square wave, its amplitude is zero). Setting the number of terms to three displays two cosine terms (both with amplitude zero) and one sine term (with some amplitude). The plot at the top of the applet now displays your function together with this sine function, the first non-vanishing term in your Fourier Sum. The approximation is quite decent already! After all, a square wave is almost a sine wave, just with sharper corners. As you include more and more terms by increasing the *Number of Terms*, you see that the cosine components always remain at zero amplitude, but that adding more and more sine terms of larger and larger frequency matches the resulting approximation, the Fourier Sum, better and better to your actual signal.

To see the effect of suppressing the lower frequency components, we will write a small Python program that will plot the resulting Fourier Sum by including terms within a certain range. This is different from the applet, which always starts at the first term and ends at the term you select with the *Number of Terms* slider. Our Python program will allow us to sum the Fourier terms starting at some arbitrary term and ending at another arbitrary one.

Open a new Jupyter Notebook and give it a good name. You will turn in this notebook together with your notes in this manual (you can share it with me via email). As always, we first have to import the correct Python modules: `numpy` and `matplotlib`. Add the following two lines to the top of your program:

```
import numpy as np

import matplotlib.pyplot as plt
```

Here, we are giving each of the modules a short-hand name (`np` and `plt`, respectively).

Next, we'll define our Fourier Sum. You can easily look it up online and confirm that the Fourier Sum for a square function that is centered at zero (i.e. no offset) is given by $V(t) = \frac{4}{\pi}\Sigma_{n=0}^{\infty}\left(\frac{1}{2n+1}\right)\sin\left(\frac{(2n+1)\pi t}{T}\right)$, where $T$ is the period of the signal, related to its frequency by $T = 1/f$, and $n$ is an integer. To type this function into Python, use the following code. Note that there has to be an indentation at the beginning of the second line!

```
def V(t, ni, nf):

    return (4/np.pi) * sum(list((1/(2*n+1)) * np.sin((2*n+1)*np.pi*t) for n in range(ni,nf)))
```

This defines our Fourier Sum as a function of time $t$ and with the two parameters $ni$ (the beginning of the summation) and $nf$ (the end of the summation). Note that here we have chosen the period $T$ to be 1 s, and thus all other times must be measured in seconds as well.

Next we need to define a range for our time variable. We can do that with the `numpy` function arange to create a discrete list of times from 0 s to 2 s in steps of 0.001 s:

```
t = np.arange(0, 2, 0.001)
```

Now we are ready to plot our function. To explore the effect of the first few terms on the overall Fourier Sum, let's show two graphs in one plot: In red, the Fourier Sum from $ni = 0$ to $nf = 50$, and in blue the Fourier Sum from $ni = 1$ to $nf = 50$. Note that we are *only* removing the very first term in the Fourier sum (i.e. we are removing the main sine term that you saw in the applet).

```
plt.plot(t, V(t, 0, 50), 'r')

plt.plot(t, V(t, 1, 50), 'b')

plt.show()
```

Execute this program and generate the final graph.

**Describe and compare the blue and the red trace. Then compare the blue trace to the signal you see on the oscilloscope when you use frequencies of less than 10 Hz. Feel free to play with the $ni$ value of the blue curve to see how the trace changes as you remove more and more of the initial terms.**

### 5.2.3 Other Channel Options:

For completeness and to act as a reference, we will briefly describe the other channel options here. The `CHANNEL MENU` has four more options: `BW LIMIT`, `VOLTS/DIV`, `PROBE`, and `INVERT`. `BW LIMIT` stands for 'Bandwidth Limit' and allows you to limit the bandwidth of your oscilloscope. The bandwidth of an instrument is characterized as the frequency width at which it operates. For example, a bandwidth of 200 MHz means that the oscilloscope takes and displays a data point with a rate of 200 MHz, or once every 5 ns. That's pretty fast! In general, that's exactly what you want; the faster the oscilloscope samples your data, the more detail it captures. Fast oscilloscopes thus allow you to display very fast data. However, if your data is also pretty noisy, especially if it has high frequency noise, then a very fast sampling rate will show all of that noise and your displayed signal looks noisy. Limiting the bandwidth in that case can lead to a cleaner signal that is displayed on the screen.

You know by now that the large knob labeled `VOLTS/DIV` adjusts the vertical scaling of the oscilloscope for each of the two channels. Selecting the `VOLTS/DIV` option in the `CHANNEL` menu allows you to change from coarse adjustment (default) to fine adjustment. Coarse adjustment allows you to change the size of a unit voltage in relatively large steps as you rotate the `VOLTS/DIV` knob. For most purposes that's fine. However, if you need to adjust a signal very precisely, then the fine adjustment will allow you to get to a very specific scale.

The `PROBE` option allows you to include external scaling factors that may be present in your system. For example, if you want to display a signal that has an amplitude of 1000 V, you cannot directly connect it to the oscilloscope since voltages that large would immediately destroy it. Instead, you need to use an external attenuator that reduces the amplitude by about a factor of 100 or so. After the attenuator, the signal now has an amplitude of only 10 V, which can easily be displayed on the oscilloscope. The `PROBE` option allows you to include this external scaling factor by setting it to '100x'. The oscilloscope will now display the actual voltage before the attenuator, even though the voltage that is sent to it is a factor of 100 smaller. Thus, the `PROBE` option does not really change your signal; it is just an option that is added for convenience, so that you don't have to convert the signal back to the original one in your head. Of course, it only works when you use standard attenuators that reduce the signal strength by 10, 100, or 1000, but those are very often commonly used. For all applications in the OPTIX lab, voltages are small enough so that they can be connected to the oscilloscope directly. So make it a habit to always check that this option is set to '1x'.

The `INVERT` option simply inverts the signal on the display without changing the slope that the oscilloscope triggers on. This can be convenient if you want to display a signal that is sent through an external inverting amplifier, but you are used to looking at the original signal and not its amplified but inverted counterpart.

If you have time, play with all of these options to make sure you understand what they do and why or when they are relevant.

### 5.2.4 `YT` vs `XY` Mode, Acquire, Cursor, and Measure:

In addition to the `YT` display mode you have used so far, in which both channels are graphed as a function of time, the oscilloscope also allows you to graph channel 2 as a function of channel 1 in `XY` mode, like a **parametric plot**. This option can be helpful if you want to compare two signals and how they correlate to one another, for example if one increases as the other increases, in which case the signal would appear as a straight light. You can get to this mode by pressing the `DISPLAY` button at the top of the oscilloscope. A lot of the options that appear next to the screen allow you to simply change the actual display of the oscilloscope in some way, for example by adjusting the contrast. Play with them to see what happens! The middle button is the one that changes the format of the graph from `YT` to `XY`. To see how this option works, split the signal from the function generator by using a BNC 'T'-connector and apply it to *both* oscilloscope channels at the same time. In `YT` mode, both channels should show you the same wave as a function of time. Now press the middle button to switch to `XY` mode.

**Sketch the wave on the screen in the box below and explain what you see.**

Right above the `DISPLAY` mode you will find the `ACQUIRE` mode, which allows you to change how the waveform data is acquired. You can chose between three options, `SAMPLE` (default), `PEAK DETECT`, and `AVERAGE`. In `SAMPLE` mode, the oscilloscope splits the horizontal axis into 2500 equidistant points, takes one reading per point, and immediately displays it on the screen. This is the mode you have used so far, and it typically works well for most signals. However, it is possible that the oscilloscope will skip over some important data points, for example if your signal changes up and down very rapidly. In contrast to the `SAMPLE` mode, in `PEAK DETECT` mode the oscilloscopes finds the minimum and maximum values during a sampling interval and displays additional values in between. Thus, the spacing is not necessarily equidistant, but is driven by the actual shape of the signal. Parts that change more rapidly from minimum to maximum will receive more data point. `PEAK DETECT` works well for lower frequencies around the order of 10 Hz, or to detect sharp pulses.

The last option is the `AVERAGE` option, which is quite useful in everyday lab life. In this mode, the oscilloscope acquires multiple samples over a sampling interval and averages them. Especially noisier signals show up much cleaner when you average over just a few samples (you can set the number of samples by pressing the button at the bottom of the options menu). Since the function generator delivers a pretty consistent and clean voltage signal, you may not see a big difference when you compare no averaging to averaging over just a few samples. However, you should see some effect when you compare the original signal without any averaging to a signal that has been averaged over 128 samples. For both signals, move the offset while watching the display.

**Set the average to 128. Then add an offset. Describe and explain what you see.**

To the left of the `ACQUIRE` menu you will see the `MEASURE` menu that allows you to measure up to five different characteristics at a time. If you have multiple signals, you can view their characteristics simultaneously. The different options should be pretty self-explanatory, so feel free to play with them to make sure you understand how to use them.

Underneath the `MEASURE` menu you will see the `CURSOR` menu. This menu is very helpful when you want to read data points off your graph. The options in this menu allow you to generate two horizontal or two vertical cursors that you can move with the `POSITION` knobs. The display on the right of the graph shows you the position of each of the two cursors as well as the difference between them. Try this!

---

And that's it! You made it successfully through this module and are now qualified to perform the more advanced modules. Please leave us any comments, suggestions, or concerns in the box below, so that we can optimize this module for future student generations. Thanks!