PHYB10 Experiment Guide

Lab 2: Alternating Voltage and Signals

Alternating current (AC) circuits have many practical applications. The electricity in homes, for example, is provided as AC. It is useful to understand how these signals can be controlled and measured.

This lab will introduce the oscilloscope and function generator. This equipment will enable the study of two important circuit components: capacitors and inductors. You will build a circuit that creates an oscillating signal from a capacitor and inductor.

The Oscilloscope

An oscilloscope is a tool that visualizes and measures rapidly alternating signals. While the oscilloscopes you will be using are digital, the controls are identical to old analogue scopes. Understanding how analogue oscilloscopes work can help you understand the settings and controls of modern oscilloscopes.

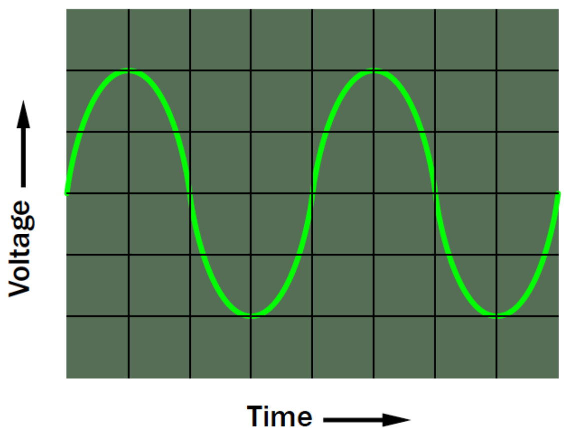


Figure 1: A simple alternating current signal.

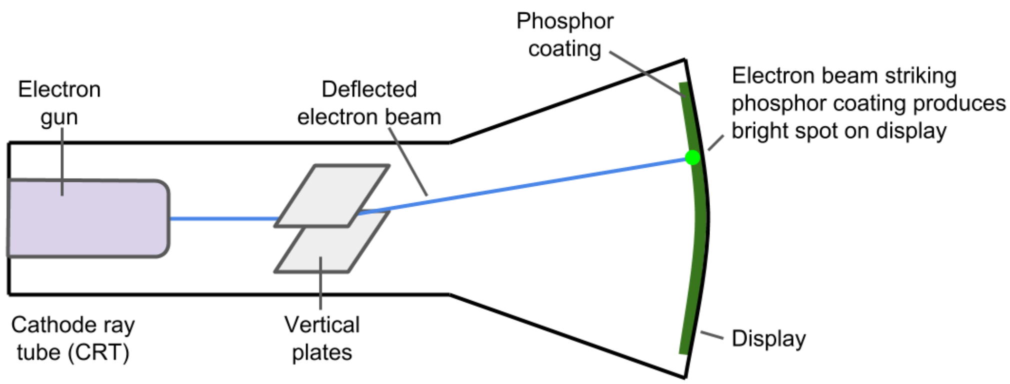


Figure 2: Experimental apparatus for analyzing the deflection of a charged particle beam using a magnetic field.

Analogue oscilloscopes were made from a Cathode Ray Tube (CRT). They fire a beam of electrons in a “sweep” from left to right on the screen, creating a solid horizontal line. The first set of controls on an oscilloscope control the timing of this sweep. You can control the speed of the sweep, i.e., how fast the beam moves left to right. Higher speeds are required to measure higher frequency signals. On modern digital scopes this is simply the scale of the x-axis. The unit of this scale is seconds/division, which is the number of seconds for the beam to move one square (1 div) along the axis. In the image above, if the scale was set to 1 s/div, the full wave form repeats every 4 div. It would then have a period of 4 s for a frequency of 0.25 Hz.

The most important control is the “trigger”. This controls when the sweep begins, e.g., “trigger the sweep”. The cathode ray tube completes one sweep and then waits for the conditions set by the trigger before starting a new sweep. This condition is a voltage level on the signal. For example, the trigger could be set to 2 V “rising”. If a signal is at 1 V the trigger will wait until the signal increases past 2 V to begin a new sweep.

The sweep of the oscilloscope and the signal both have their own characteristic frequencies. The trigger coordinates these two to create a consistent, stable image on the screen. Consider an oscilloscope that began a new sweep every ½ second (frequency of 2 Hz) and you were trying to measure a signal with a frequency of 3 Hz. Without the trigger the image on the screen would cycle and appear to change at a frequency of 1 Hz. Instead with the trigger, the image will reset itself every sweep and you will have stable image.

Modern oscilloscopes still have a trigger, and the purpose is somewhat the same. While a modern digital oscilloscope does not “sweep” its electron gun and could theoretically sample the input signal indefinitely, its screen still has a refresh rate. The trigger coordinates the refresh frequency of the screen and the signal to maintain a stable image.

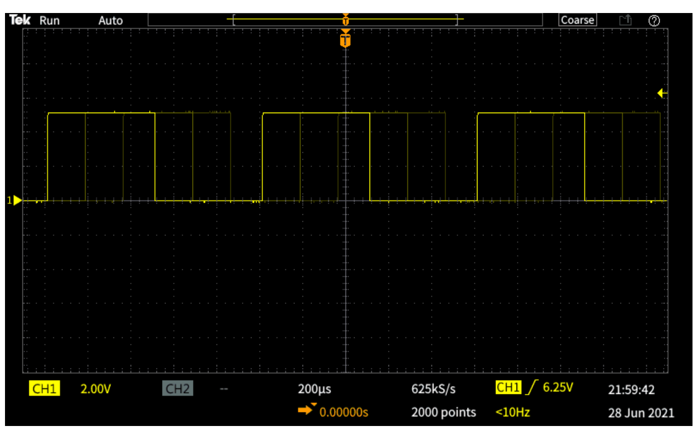
The y-axis of analogue oscilloscope was controlled by a pair of charged plates that are attached to the signal you wanted to measure. The charge on these plates would create a deflection of the election beam. For example, if the top plate was positively charged the beam would deflect upwards and hit the screen higher up. The scale of the y-axis could be controlled with an amplifier. A small signal could then be expanded to fill the whole screen. The units for this are V/div. If the scale was set to 1 V/div in the image above, the signal would have an amplitude of 4 V peak to peak (4 V pk-pk).

Exercise 1: Oscilloscope and Function Generator Basics

Step 1: Turn on the oscilloscope and attach a probe to channel 1. The alligator clip on the probe is the reference voltage and is typically attached to GND or 0 V on your circuit. The tip of the probe can be clipped onto a component to measure the signal.

On the bottom right of the front panel there is a metal loop called “Probe Comp”. Attach the alligator clip to the bottom loop and the tip to the top.

Turn on channel 1 by pressing the “1” button outlined in yellow under the vertical control section. You should now see a square wave on the screen, although right now it might be hard to see.

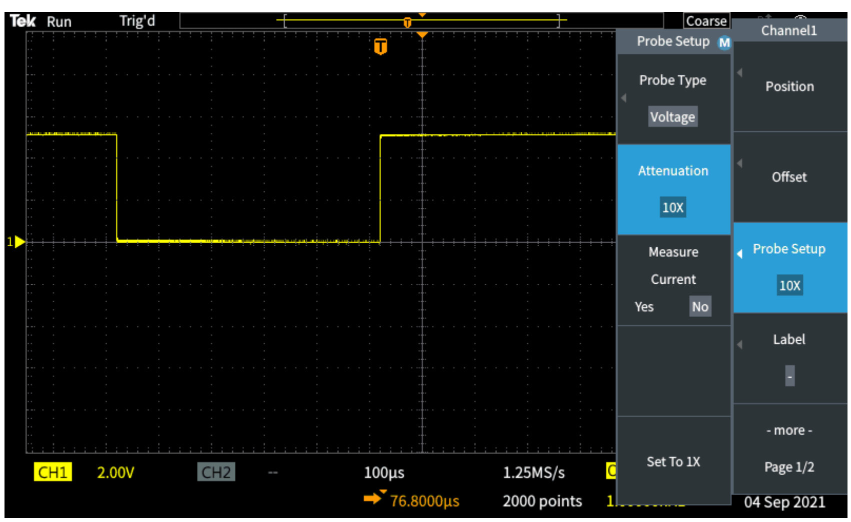


Step 2: Look at the connector on the probe. You will notice it has a marking labeling the probe as "10×". This is because the probe has a built-in resistor that will reduce the signal by a factor of 10 times. The probe includes this resistor to lower the error due to input impedance you learned about in the last lab. However, <u>we must tell the oscilloscope to compensate for this attenuation</u>.

Open the channel 1 menu by pressing the "1" button. Find the "Probe setup" window, press the adjacent button, and set "Attenuation" to "10×".

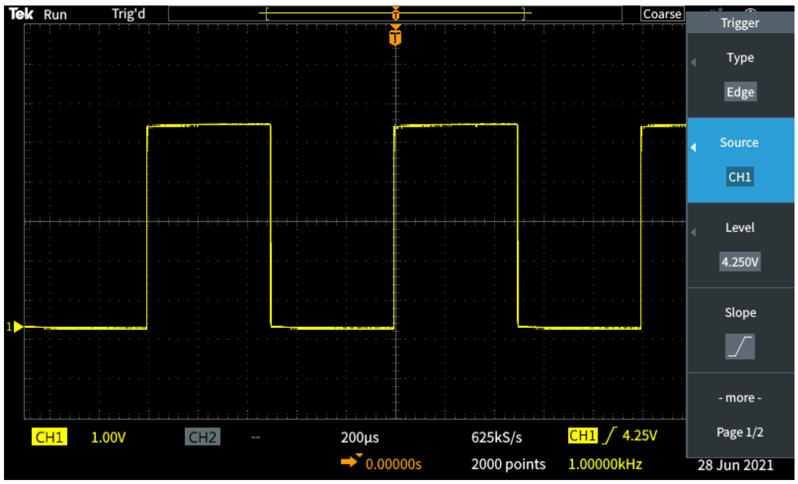
If a regular cable is plugged into the channel the attenuation should be set to 1×.

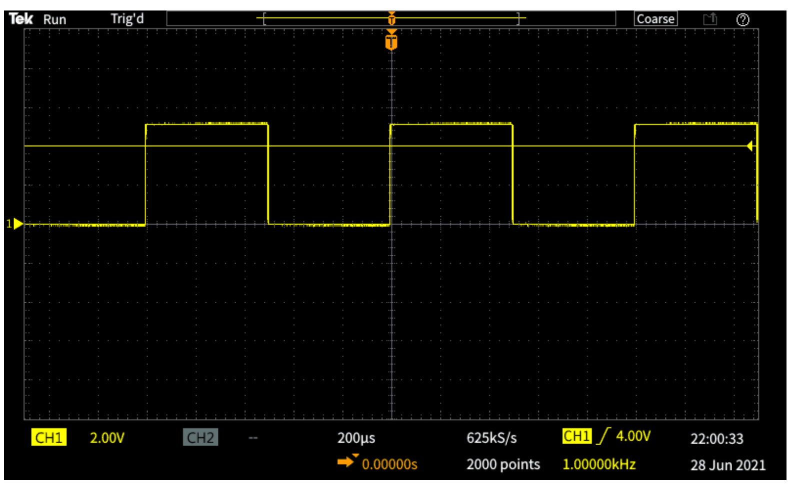
Always ensure the attenuation is set correctly! This is a difficult error to catch. The waveform will look correct on the screen but all your measurements will be off by a factor of 10.



Step 3: When analyzing a signal, you must first adjust the trigger level. Trigger controls can be found on the far-right side of the oscilloscope control panel. Press the “Trigger menu” and the trigger options should appear on screen. In the “source” section it should read “CH1” (channel 1). If not, press the adjacent button and cycle through your options until it does. Then, use the knob to adjust the trigger level. While you are rotating the knob, a parallel line should appear on screen. This is the trigger level. It must intersect the input waveform to work.

Adjust the trigger level until you have a stable image on your screen.

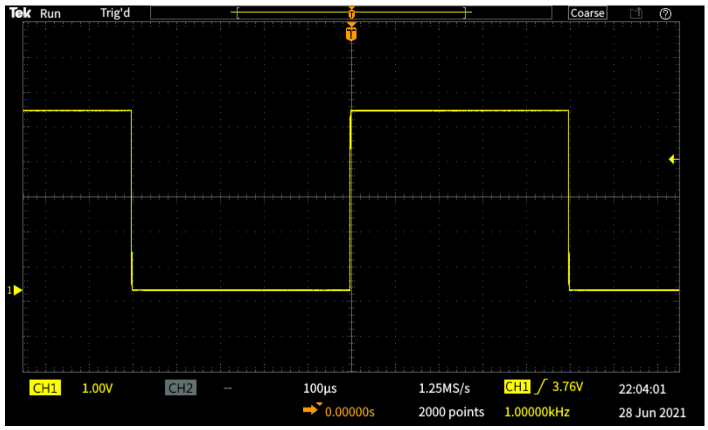


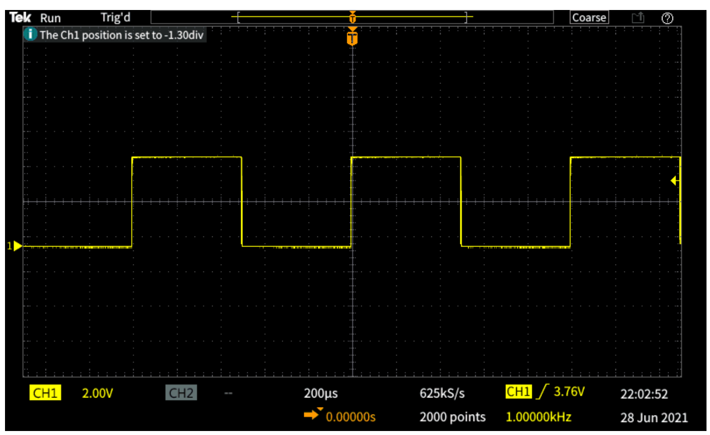


Step 4: The scales of the axis can be adjusted using the appropriate knobs in the vertical and horizontal controls. The position knobs control where the waveform appears on the screen. When viewing two waveforms, you may want to have channel 1 located towards the top of the screen and move channel 2 towards the bottom.

What you want to view depends on what you are measuring.

At the bottom of the screen, the scale selected and the waveforms position away from center will be displayed. Use the position knobs to center the waveform and scale to have 2 – 3 wavelengths on screen and amplitude a large as possible. You want to be able to see the whole wave form.

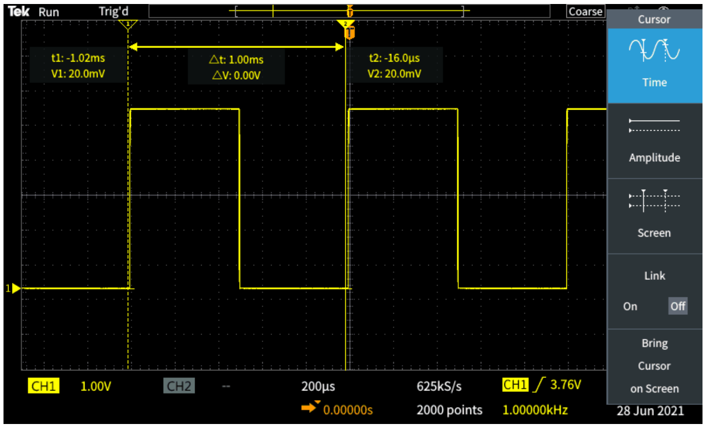




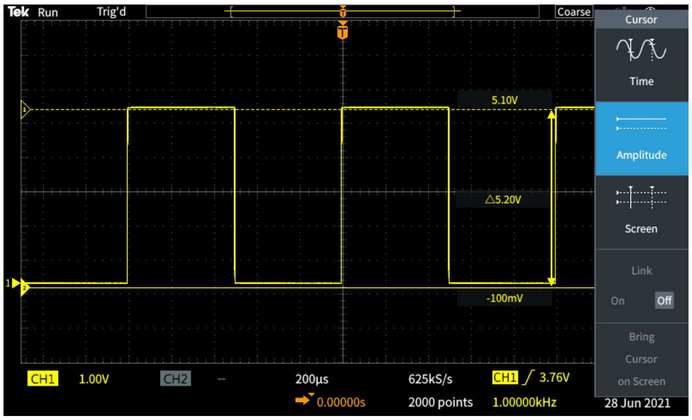
Step 5: Modern oscilloscopes come with tools that allow for high precision measurements to be taken. We will be using two of these tools in this course: cursors and the measure window.

First let's trying using the cursors.

Beside the multipurpose knob is the cursor button. Press this button to turn on the cursors and menu.



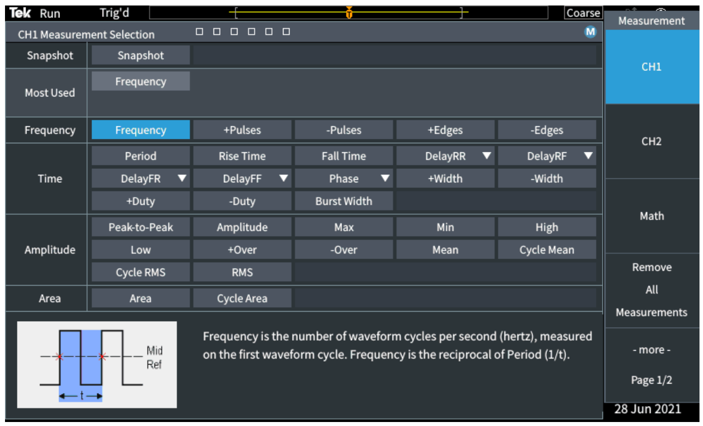
You can select the type of cursor using the buttons beside the screen. The cursors are moved using the multipurpose knob. The cursor selected can be changed by pressing the multipurpose knob (like a button). The cursors show the absolute value of its location as well as the difference between positions. The difference is useful for calculating period or amplitude, depending on the cursor used.

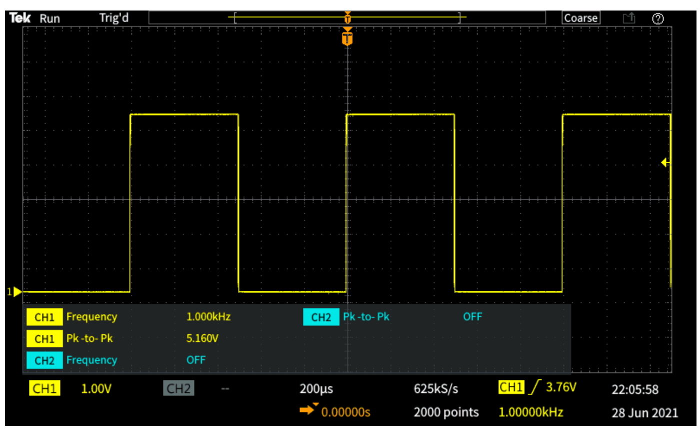


The oscilloscope can also be set up to take automatic measurements. Press “measure” to open the menu. From this list you can select up to 6 measurements to be displayed on screen.

For this course the most useful will be frequency and peak-to-peak amplitude (pk-to-pk).

Select CH1 from the buttons beside the screen and then use the multipurpose knob to select frequency and pk-to-pk. Repeat this with CH2. Press the “Menu On/Off” button to close the measurement menu. You should now see your measurements at the bottom of the screen.





Step 6: Set up the function generator for use as an input to the circuits.

Attach a T connector to the output of function generator and then attach one end of connector to channel 2 of the oscilloscope using a BNC cable.

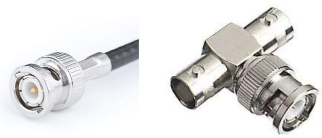
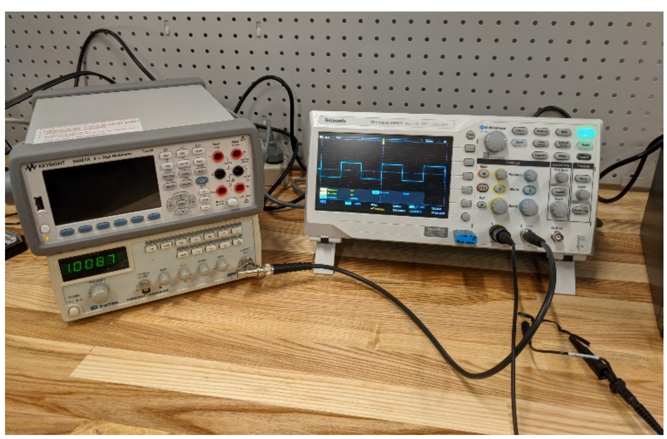


Figure 3: Left: BNC cable. Right: BNC T-connector.

Turn on the function generator and set it to square wave (look for button with an image of a square waveform). Turn on channel 2 of the oscilloscope by pressing the “2” button in the vertical controls. Make sure the attenuation on this channel is set to “1×”.

Adjust the frequency and amplitude of the function generator until it is identical to the waveform on channel 1.

The amplitude is adjusted using the “AMPL” knob. You may find it easier to get the correct amplitude if the attenuation button “ATT –20dB” to the right of the waveform selection is also on.



You may notice that channel 2 is not triggering properly. The oscilloscope can only be triggered to on channel at a time. Since the timing of the “Probe comp” wave form and the function generator are not coordinated it will be impossible to get a stable image of both waveforms at the same time. Turn of CH1 by pressing the “1”, then use the trigger menu to set the trigger to CH2. Adjust the trigger level until you have a stable image.

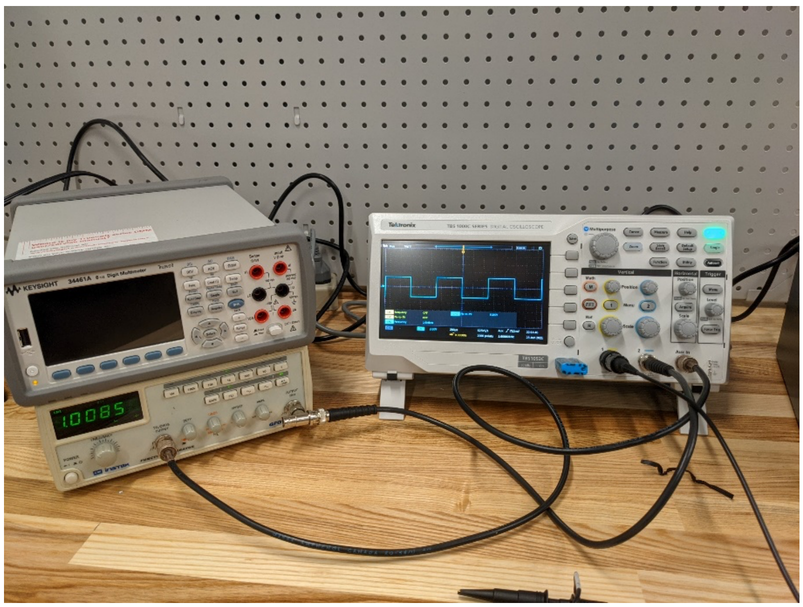
Step 7: Many different circuits are used in this course. It will be useful to use the function generator as an external trigger source because our waveforms on channel 1 and 2 will be changing. The function generator has an output that is a consistent square wave with constant amplitude, but the frequency will match the main output's settings. This is the "CMOS" output.

Attach the CMOS output of the function generator to the "Aux in" input of the oscilloscope using a BNC cable.

Open the trigger menu and change the source to "Aux in".

Navigate to the 2^{nd} window of the trigger menu and change "Probe comp" to 1×.

You can then adjust the trigger level to this aux signal.

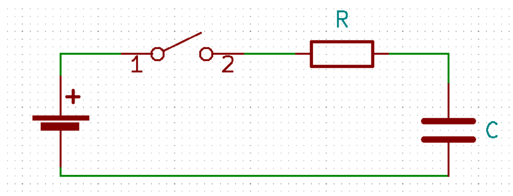


Capacitors

Capacitors are a very useful tool in circuit design. The simplest experiment that can be done with them is to examine their behaviour when a constant DC voltage is applied to charge them. Using a capacitor as source of charge, somewhat like a battery, does not demonstrate their full potential for practical use. In this section we will explore the properties of a capacitor and how they can be used to process AC signals. This can be useful when building sensors.

DC Voltage Review

The circuit below contains a battery, switch, resistor, and capacitor:



When the switch is closed, the capacitor will begin charging. The voltage around the loop is given by the equation:

$$V_{bat} + V_c + V_R = 0$$

The voltage across the resistor can be found using Ohm's Law. The voltage across the capacitor is:

$$V_c = \frac{Q}{C}$$

where Q is the current charge on the capacitor and C is its capacitance. Substituting this into the previous equation results in:

$$V_{bat} + \frac{Q}{C} + IR = 0$$

The current can be written as the rate of charge flow to or from the capacitor, yielding the equation:

$$V_{bat} + \frac{Q}{C} + \frac{dQ}{dt}R = 0$$

This equation can be solved for Q and then substituted back to V to obtain:

$$V_c(t) = V_{bat}\left(1 - e^{-t/\tau}\right)$$

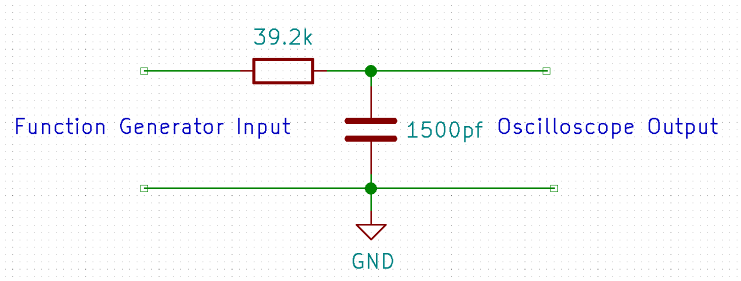
with the time constant $\tau = RC$.

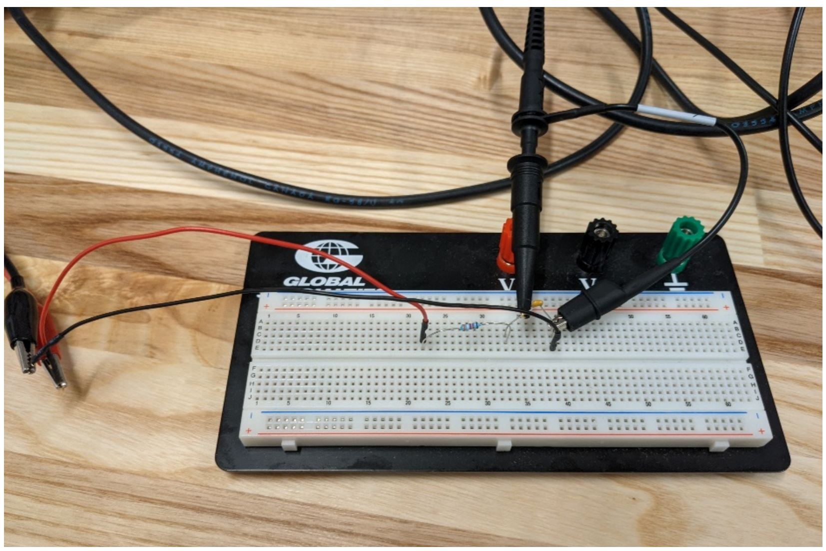
In the following experiment we will be studying a similar circuit. Instead of a battery and a switch, we will be using a function generator set to a square wave. This square wave works like a switch and battery. When square function is at its maximum, this is like the switch being open. When the function is at a minimum is like the switch being closed. The frequency of the function is how fast the switch is being opened and closed.

Exercise Two: Capacitors DC to AC operation

Step 1: Build the following circuit. Use a BNC to alligator clip cable and attach to the free end of T connector on the function generator. Use jumper cables to attach the generator to the circuit.

The other side of the T connector should remain attached to channel 2 of the oscilloscope. Attach the channel 1 probe to the capacitor.





Step 2: Turn on the function generator and set to a square wave with an amplitude of 1V pk-pk and a frequency of 200Hz. Use the measurement tools of oscilloscope to help you do this.

Question 1: [2 points]

From your observation of this waveform, would you say the capacitor is fully charging and discharging on each "switch" of the square wave? Why or why not?

Step 3: Increase the frequency of the input square until the capacitor is no longer fully charging.

Question 2: [2 points]

At what frequency can the capacitor no longer charge or discharge? How does the period of this input waveform relate to the time constant of the resistor and capacitor? Take at least 10 measurements of the amplitude of the output waveform at input frequencies starting at 200Hz and doubling for each new measurement. Create a graph of Amplitude vs. Frequency using software on a computer. Use a logarithmic scale on the x-axis.

Exercise 3: Capacitors and AC operation

Step 1: Keep the same circuit and setup as Exercise 2.

Step 2: Turn on the function generator and set to a sine wave with an amplitude of 1 V pk-pk and a frequency of 200 Hz.

Question 3: [2 points]

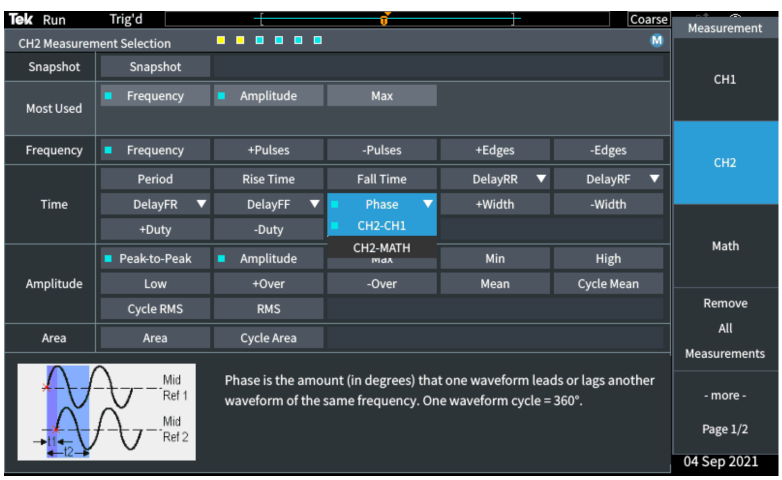
Similar to the last exercise, take at least 10 measurements of the amplitude of the output waveform at various input frequencies, starting 200 Hz doubling for each measurement. Create a graph of your results.

Question 4: [4 points]

In Exercise 2 the output waveform appeared very different to the input square wave. In this exercise both the input and output waveforms are sine waves. However, you may notice that the phase of the output waveform changed depending on the input frequency. i.e., the peak of output waveform was “delayed” from the input frequency. You can measure this change using a measurement tool on the oscilloscope.

Open the “Measure” window and select “Phase” and “CH2-CH1”. Measure the phase change of the output using the same frequencies as the last question.

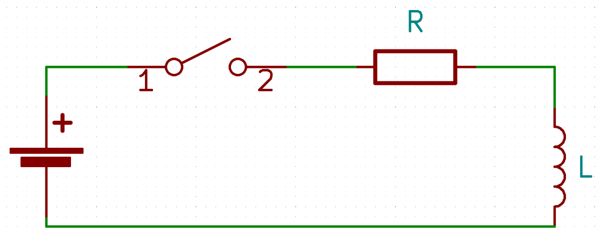
Create a graph of phase change vs. frequency. [2 points]



Although the waveforms looked very different in Exercise 2 do you think there was a similar phase change when using the square wave? From your knowledge of capacitors, hypothesize what is causing this phase change. [2 points]

Inductors

Inductors are useful in sensor design because they create magnetic fields. These magnetic fields can interact with objects outside of the inductor and change how the inductor behaves in a circuit. To understand how sensors with inductors work we first need to understand how a regular inductor behaves in a circuit. Let's start with the circuit below:



The voltage around this circuit is given by the equation:

$$V_{bat} + V_L + V_R = 0$$

The voltage across the resistor is given by Ohm's law and across the inductor is given by:

$$V_L = L\frac{dI}{dt}$$

Substituting this all in we have:

$$V_{bat} + L\frac{dI}{dt} + IR = 0$$

This is a differential equation. We can solve for I and then substitute to get back to an expression for V:

$$V_L(t) = V_{bat}e^{-t/\tau}$$

with the time constant $\tau = \frac{L}{R}$.

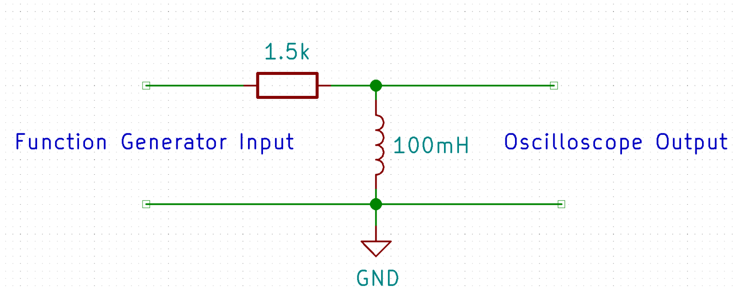
Note: even though the mathematics of capacitor and inductors look very similar, their behaviour are opposite.

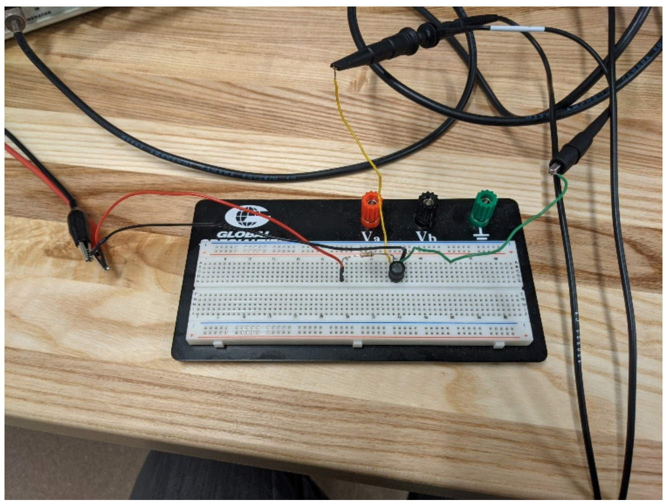
- For the capacitor circuit: When the switch is turned on, there is a large initial current, but as the charge on the capacitor builds up, the current is repelled by this charge build up. The voltage across a capacitor starts at zero and builds up to a steady state as charge accumulates on its plates. The current starts high and eventually goes to zero.
- For the inductor circuit: When the switch is turned on, there is a large change in the current. This starts to build a magnetic field in the inductor coil. However, this magnetic field repels the charges in the current through mutual induction and generates a large voltage in the opposite direction of the current flow. The current stats at zero and eventually builds up to a steady state. The voltage is high initially but goes to zero over time.

We will now study how the inductor circuit behaves when a sine signal is input. We will then compare the results to the capacitor circuit to see if the "opposite" behaviours also appear.

Exercise 4: Inductors

Step 1: Build the following circuit. Use jumper cables to measure the voltage across the inductor.





Step 2: Turn on the function generator and set to a sine wave with an amplitude of 1 V pk-pk and a frequency of 200 Hz.

Question 5: [2 points]

Similar to the capacitor exercises, take at least 10 measurements of the amplitude of the output waveform at various input frequencies, starting 200 Hz doubling for each measurement.

Take measurements of the phase change at the same time.

Create graphs of output amplitude vs. frequency and phase change vs. frequency.

AC Signal Processing – Filters

You should have graphs showing the output when inputting sine waves in the following circuits:

- Capacitor-Resistor Circuit: Output Amplitude vs. Input Frequency; Output Phase Change vs. Input Frequency
- Inductor-Resistor Circuit: Output Amplitude vs. Input Frequency; Output Phase Change vs. Input Frequency

These circuits have very different behaviours but both can act as filters.

Filters eliminate or "attenuate" (reduce amplitude) of specific frequencies from a signal. Because the attenuation is determined by the time constant of the circuit, filters can be designed to eliminate unwanted frequencies from a signal. Unwanted frequencies in a signal you are analyzing is called "noise". For example, there is often a 60 Hz signal in any electronic device. This is because the electricity grid operates at 60 Hz. The electric and magnetic fields of that grid are all around us. If we want to look at a signal from a source that is not the grid it is useful to use a filter to eliminate that 60 Hz noise.

Question 6: [2 points]

Engineers use a shorthand vocabulary to describe the effect of filter. A "Low-Pass" filter allows lower frequencies to pass and attenuates higher frequencies. Conversely, a "High-Pass" filter allows higher frequencies to pass and attenuates lower frequencies. Which circuit(s) in this lab (Capacitor – Resistor or Inductor – Resistor) would you describe as a low-pass filter and which is a high pass filter? Reference and relate this to the graphs you created.

ADCs and Aliasing

Another common use for low pass filters is before an Analogue to Digital Converter (ADC). An Analogue to Digital Converter was used in Lab 1 when making a digital thermometer. An ADC converts any signal from a sensor to a digital number that can be used by a computer. You may recall that the output of the digital thermometer was a steady stream of temperatures. The ADC continuously takes measurements at a rate called a "sampling frequency". The sampling frequency for an Arduino ADC is 9600 Hz.

There is an error called "aliasing" that will occur if an ADC tries to measure a signal that has a higher frequency than the ½ the sampling frequency. This is called the Nyquist Frequency. When designing electronic sensors, it is often easier to place a low-pass filter before the ADC to eliminate any frequencies above the Nyquist to avoid aliasing.

Question 7: [2 points]

What is the Nyquist frequency for an Arduino? Examine your graph of output amplitude vs. frequency for the low-pass filter and estimate the lowest frequency where there would be complete attenuation. Would the filter we built be suitable for eliminating aliasing for our Arduino? Would you need a circuit with a higher or lower time constant based on the Nyquist frequency of the Arduino? Hypothesize a circuit with a time constant and values for components that would work for this purpose.

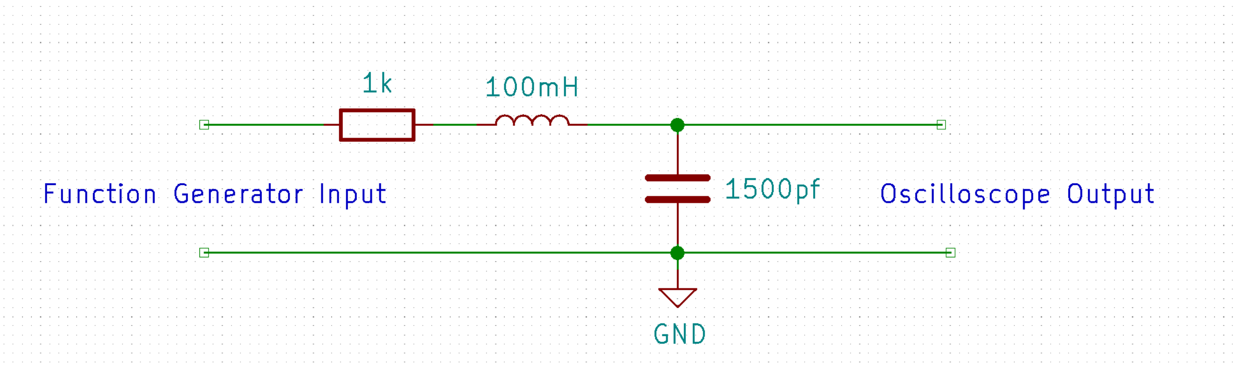
Filters are a very important topic in sensors and electronic design. Another method for building filters using Operational Amplifiers is examined in Lab 4.

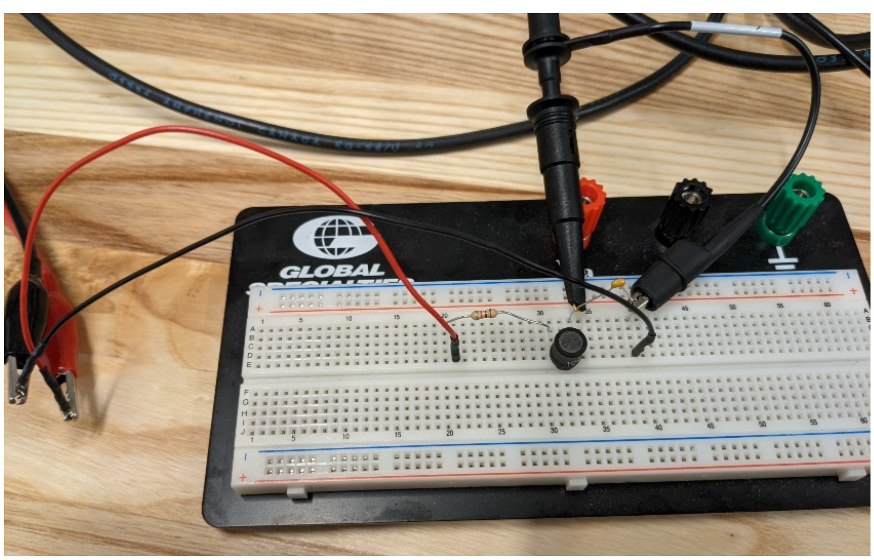
RCL Band-Pass Filter or Resonator Circuit

We have studied a low-pass filter and a high-pass filter. We will now combine these into a band-pass filter using a Resistor-Capacitor-Inductor (RCL) circuit. If these components are selected correctly, the capacitor will attenuate high frequencies, the inductor will attenuate low frequencies, and what is left is a narrow band of frequencies that are allowed to pass. Further, if these components are selected extremely carefully, the circuit will output a single sine wave at one frequency. This is why the circuit is called a “resonator circuit”.

Exercise 5: RCL resonator circuit as a Magnetic Field detector

Step 1: Build the following circuit. Measure the voltage across the capacitor using the oscilloscope.





Step 2: Turn on the function generator and set it to produce a square wave of 200 Hz and 5 V pk-pk. Take a screen shot of the oscilloscope by inserting a USB drive and pressing the save button.

Question 8: [2 points]

Describe the output wave form of the RCL circuit. Include a screenshot from the oscilloscope. Would you describe it as a resonator?

Question 9: [2 points]

Try holding a piece of metal near the inductor like a coin or keys. What happens to the output wave form? If you would like to see an even bigger impact ask the lab technician for a small magnet and hold this next to the inductor.

Why does the circuit behave differently near metal or a magnetic field? How could you use this property of the circuit to turn this circuit into a metal detector or magnetic field sensor?