ECE20007: Experiment 7

Passive Filters

Ryan Beasley, Ben Manning, Purdue University
Conner Hack, Editor
Last modified: October 18, 2023

---

1 Application

Previously we looked at RC circuits in a transient state. While these circuits are useful for controls and timing, they are not the only way to use RC, RL, or RLC circuits. Another useful function of complex impedance circuits is a filter. Just like a mechanical filter removing unwanted material, an electrical filter removes unwanted signals.

There are many different type of filters, but they all can be described as one of four categories.

- Low-Pass Filter (LPF)
- High-Pass Filter (HPF)
- Band-Pass Filter (BPF)
- Band-Stop Filter (Notch Filter)

Filters have many applications in the real world and all of them allow your devices to function the way you intend. Be it the filters on the input of your Wi-Fi receiver removing the FM signal from your local radio station, or the high-pass filter that is on the output of the switching power supply powering your AD2 suppressing the 60Hz line frequency of the wall outlet.

At times during your engineering career, you will find that all systems you work with will not be ideal and you will need to work around that as best you can. Sometimes the non-ideal factor of a system can be ignored, or sometimes it works in the engineer's favor, but most of the time you will need to correct for it. A simple way to clean up your signals is with a filter. With filters you can control the frequency of signals that will be allowed to ODO High-Pass Filter 6.pass into the rest of your system. Filters can be either passive or active, but in this experiment we will be looking only at passive filters.

2 Experiment Purpose

This experiment will explore one specific use case (and a very useful one, at that) of capacitors and inductors. Series and parallel combinations of resistor, capacitors, and inductors will produce passive filters that make circuits exponentially more useful in real world application. Filters can be described very precisely using differential equations, and we will walk you through those calculations during the reading, but the focus of the experiment will be on the implementation and characteristics of the filters.

By the end of this experiment you should be able to recognize the various filter topologies within circuits and feel comfortable explaining what their function is within a system.

3 Reactance

In the timing experiment we mentioned that both capacitors and inductors are reactive components. If you remember from there we mentioned that reactive component "react" to the current state of the circuit. As the circuit changes (i.e. currents at certain nodes fluctuate), the component will behave differently.

With the addition of capacitors and inductors, our circuits become much more involved in construction and calculations. Because these devices are reactive, you will see that their behavior is frequency dependant. If we look at a typical voltage divider, we can replace either R_1 or R_2 with a capacitor or inductor and we will get a frequency dependent voltage divider.

3.1 Capacitor Reactance

In an alternating current circuit, the current alternates. Because of this, the rate of movement of charge (current) through a capacitor is proportional to the voltage, frequency, and capacitance. If we take the effect of capacitance and frequency considered together, they form a quality similar to resistance. We call this quality **capacitive reactance** and it has the unit of ohms, similar to resistors. The formula for caluclating capacitive reactance is:

$$X_c = \frac{1}{2\pi fC} = \frac{1}{\omega C} \tag{1}$$

Where X_c is the capacitve reactance in Ohms, f is the frequency in Hz, C is the capacitance of the capacitor in farads, and π is π. We will often work with ω, or the angular frequency, instead of $2\pi f$ to simplify some calculations. Now that we understand how to calculate reactance of a capacitor, but what is that really telling us? Let's see if we can break this down a little more. If we focus on a sinusoidal voltage source, let's say it's $V_o\cos(\omega t)$, we can plug that equation into the expression for the displacement current for a capacitor. If you remember, $I = CdV/dt$ is that expression. We end up with something like this:

$$I = C\frac{dV}{dt} = -\omega CV_o\sin(\omega t) \tag{2}$$

Because the current flowing through the capacitor is proportional to the rate of change of the voltage across that capacitor, we can calculate the instantaneous current at any point for an AC voltage across the capacitor. While this is possible, it is not really the most useful use case for this principle. If we instead focus on a maximum voltage and current, it will make our calculations and conclusions a little more clear. Our max current, I_o, will occur when $\sin(\omega t) = -1$, that will mean $I_o = \omega CV_o$, to simplify the calculations. The ratio of peak voltage to peak current V_o/I_o resembles a resistance, but unlike the resistance we are used to dealing with in resistors, this does not produce heat. Because of this main difference we refer to this physical phenomenon as reactance and not resistance.

Referring back to Equation 1, we can see that as frequency approaches ∞, X_c approaches 0. Because of this the capacitor will act as a short at high frequencies. With that knowledge and the knowledge we have of capacitors at DC voltages, we can also see that as the frequency approaches 0, X_c approaches ∞ . At that point the capacitor will act as an open circuit to the rest of the circuit.

Note: Yes, reactance has the unit of Ohms, which allows us to treat our reactive components as resistors, at least temporarily for calculated frequency. This means that we can use Ohm's Law, Kirchhoff's Laws, and even our voltage divider equations to determine how a circuit will act at a specific frequency. As frequencies change, we will have to recalculate reactance, but once we do that, we are back to our fundamentals. If you are monitoring a circuit at multiple frequencies (discussed below), it will be worth your time to simulate the circuit using Spice, or your preferred programming language, such as MATLAB or Python.

3.2 Inductor Reactance

The inductor's rate of movement of charge (current) through it is inversely proportional to the voltage, frequency, and inductance. In a very simple case, we can think of the inductor as the "opposite" of a capacitor. Where a capacitor wants to maintain the voltage across it the inductor is trying to maintain the current through it. This is due to the current inducing a magnetic field and that magnetic field inducing currents.

Just like with the capacitor if we take the effect of inductance and frequency together they also give us a qualitity similar to resistance called **inductive reactance**. The formula to calculate inductive reactance is:

$$X_l = 2\pi fL = \omega L \tag{3}$$

Where X_l is the inductive reactance in Ohms, f is the frequency in Hz, L is the inductance of the inductor in henries, and of course π is π. Again sometime we will work with ω instead of $2\pi f$ to make calculations easier.

To derive the expression for inductive reactance we first need to connect the indcutor to a sinusoidal voltage source. To make the integral a little more friendly, we are going to use a cosine function instead of a sine function. So our source voltage is $V_o \cos(\omega t)$, The current going through the inductor is:

$$I = \frac{1}{L}\int V dt \tag{4}$$

Replacing V with our source voltage we get:

$$I = \frac{1}{L}\int V_o \cos(\omega t) dt = \frac{V_o}{\omega L}\sin(\omega t) \tag{5}$$

Plus a constant, but we are assuming that constant is 0.
Similar to how we evaluated the capacitor, the peak current (I_o) occurs when $\sin(\omega t) = 1$. Then

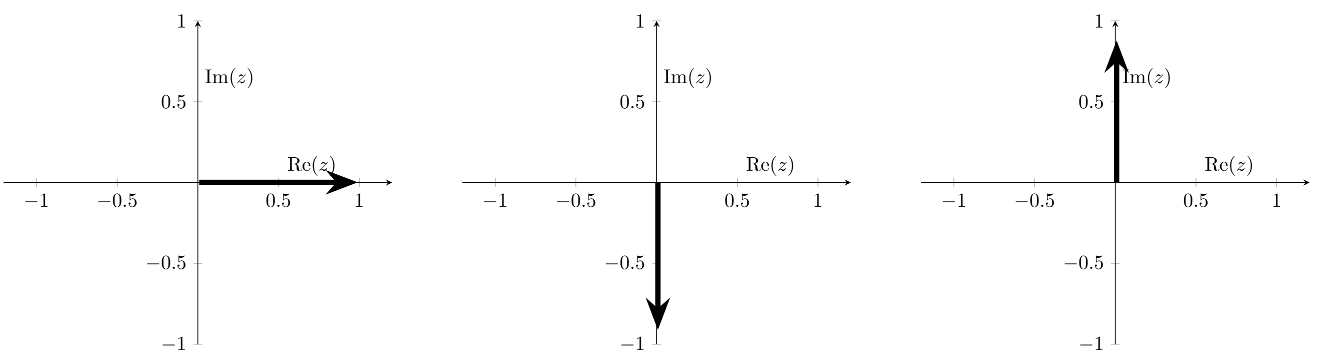
$$I_o = \frac{V_o}{\omega L} \tag{6}$$

Finally, if we rearrange this equation, you end up with:

$$\frac{V_o}{I_o} = \omega L = X_L \tag{7}$$

4 Impedance

To assist in solving AC circuits, we are introducing a new concept called impedance. Just as resistance is the opposition of current in DC circuits, impedance is the opposition of current in AC circuits. Impedance is a complex value with both real and imaginary elements. Looking at impedance on a complex numbers plain we can see that a ideal resistor is purely in the real plane and both ideal inductors and ideal capacitors are purely in the imaginary plane.



(a) Resistor Resistance in the Complex Plane

(b) Ideal Capacitor Reactance in the Complex Plane

(c) Ideal Inductor Reactance in the Complex Plane

The concept of impedance is just "generalized resistance" to help us deal with the way capacitors and inductors affect a linear system. Impedance is:

$$\boldsymbol{Z} = R + jX \tag{8}$$

Where Z is the impendance, R is resistance, X is reactance and $j = \sqrt{-1}$. Notice the j in front of the impendance. This shows that once introduced to Ohm's Law, the phase of the voltage and current will be 90° out of phase. While j is an imaginary number, it does not actually represent "imaginary" voltage or current, it just allows us to properly model the behavior of the components with the math we have available by making the imedance into a vector with a real and complex part. If you are interested in learning why j is used to model reactive components, do some research in Maxwell's equations, which involves second-order differential equations.

Fun Fact: Complex vectors like these are used to model other physical components too. Sound vibrating in a pipe different holes throughout it creates a complex waveform which needs complex numbers to be properly modeled. Flutes, especially in high octaves, need to be modeled using complex waves in order to fully understand what is going on with the sound.

Throughout your time as an engineer you will hear phrases like "output impedance is 50Ω" or "adjust the waveform generator to high impedance mode." The reason we use the word impedance is it covers both resistance and reactance.

Because we know that all real components are not ideal, we need a way of discussing the effect components have on an AC circuit that takes both resistance and reactance into account. Many of the techniques we have developed while working with DC circuits carry over to AC circuits when we look at impedance.

Using this information, we can treat impedance in an AC circuit similar to how we treat resistance in a DC circuit. So then Ohm's Law becomes

$$\mathbf{V} = \mathbf{I} \times \mathbf{Z} \tag{9}$$

The bold values indicate vector values As indicated above, Ohm's Law holds for AC circuits with the introduction of impedance. Because Ohm's Law holds, that means that you can add impedance in series and in parallel, just like resistors in DC.

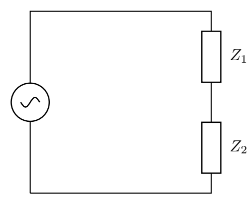


Figure 2: Two impedances in series

In Figure 2 the total impedance would be

$$Z_{total} = Z_1 + Z_2 \tag{10}$$

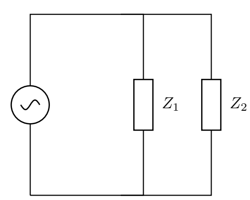


Figure 3: Two impedances in parallel

In Figure 3 the total impedance would be:

$$Z_{total} = Z_1 || Z_2 = \frac{Z_1 Z_2}{Z_1 + Z_2} \tag{11}$$

4.1 Capacitor Impedance

If we breakdown the impedance of a real capacitor into its real and imaginary components, we get the resistance (real) and the reactance (imaginary) vectors that describe the component fully.

$$Z = R_c + jX_c \tag{12}$$

This equation will get us the full impedance of our capacitor. This includes the equivalent series resistance (**ESR**).

$$Z = R_c - j\frac{1}{\omega C} \tag{13}$$

While all capacitors have a series resistance (refer to as ESR) during this experiment, you can treat R as zero and focus on the reactance of the capacitor.

$$Z = \frac{-j}{\omega C} = \frac{1}{j\omega C} \tag{14}$$

This may not hold with more complex circuits, but will be a good approximation for our calculations. **Notice that the sign changed in the complete equation for the impedance of a capacitor.** This is because the capacitor introduces a -90° phase shift between the voltage and current. Equation 14 also shows that we can manipulate the impedance equation to work towards our advantage. As long as you remember your imaginary number equations, we can make calculation easier.

4.2 Inductor Impedance

Similarly, if we break down the impedance of a real inductor into its real and imaginary components, we get the resistance (real) and the reactance (imaginary) vectors that describe the component fully.

$$Z = R_l + jX_l \tag{15}$$

This equation will get us the full impedance of our capacitor. This includes the equivalent series resistance (**ESR**).

$$Z = R_l + j\omega L \tag{16}$$

While all inductors have a series resistance (refer to as ESR) during this experiment, you can treat R as zero and focus on the reactance of the inductor. This will not hold with more complex circuits, but will be a good approximation for our calculations.

5 Decibels

We have found that it is convent to discuss relative amplitudes between two signals. For example, if an amplifier has an output voltage that is 10 times the input voltage, we can set up the ratio:

$$\frac{V_{out}}{V_{in}} = \frac{10\mathrm{V_{AC}}}{1\mathrm{VAC}} = \quad 10 \tag{17}$$

If this ratio (V_{out}/V_{in}) is greater than one, we say that the circuit has **gain**. If there is a circuit whose output is less than the input, then the gain ratio will be less than one.

$$\frac{V_{out}}{V_{in}} = \frac{1\mathrm{VAC}}{10\mathrm{VAC}} = \quad 0.1 \tag{18}$$

Because this ratio is less than one, we say the circuit has **attenuation**.

While dealing with ratios that are similar, orders of magnitude works out pretty well. You will find that often you will be working in scales that are much wider that will comfortably fit on a linear graph. Because of this, when discussing power ratios, we tend to shrink the working scale down by fitting them on a logarithmic scale. This is what we call a decibel.

To understand the decibel we first need to understand what a **bel** is. The bel is defined as the logarithm of a power ratio and gives us a way to compare power levels. The bel is defined as:

$$\mathrm{bel} = \log\left(\frac{P_1}{P_0}\right) \tag{19}$$

Where P_0 is the reference power and P_1 is the power you are comparing with the reference power.

We often use the bel to compare electrical power levels, though it is more common to use the decibel (dB). A decibel is one tenth of a bell and we use the equation.

$$\mathrm{dB} = 10\log\left(\frac{P_1}{P_0}\right) \tag{20}$$

There are a number of power ratios you will learn to recognize, but the one you will need for today is -3dB. The -3dB point is where a signal has half the power of the reference signal. We can prove this by doing the calculations. If we have two signals, and our signal's power is half that of our reference signal, then we get:

$$\text{dB} = 10\log\left(\frac{1}{2}\right) = -3.01\text{dB} \tag{21}$$

The negative sign indicates a decrease in power, or attenuation. For the -3dB point, we ignore the fraction and just refer to it as -3dB.

The -3dB point is relative to the pass band power. While we are discussing passive filters with no positive gain, the -3dB point will be at -3dB compared to system with 0dB gain. Once we start discussing active filters and gain that will change. Because the -3dB is relative to the pass band gain, it could be any value. If the overall gain of the filter is anything other than 1, the -3dB point will be $X\text{dB} - 3\text{dB}$, with X being the gain of the pass band.

While this is useful while discussing power. Typically we want to discuss output vs. input voltages. To do this, we make a rather large assumption: we assume the $R_{in} = R_{out}$. While this assumption seems unreasonable at times for most applications we want to transfer the most power to the system. We (engineers) know that the "Maximum Power Point" is when input impedance matches output impedance. So:

$$\begin{aligned} \text{dB} &= 10\log\left(\frac{P_1}{P_0}\right) \\ &= 10\log\left(\frac{V_{out}^2/R_{out}}{V_{in}^2/R_{in}}\right) \\ &= 10\log\left(\frac{V_{out}}{V_{in}}\right)^2 \\ &= 20\log\left|\frac{V_{out}}{V_{in}}\right| \end{aligned} \tag{22}$$

This relationship does not hold when input and output impedances do not match, but to make life a little simpler we assume that it does when describing a system, such as a filter.

6 Passive Filters

An electrical filter is a system that will allow certain frequencies of signals to pass while stopping others. The frequency ranges that pass through the filter is called the **pass band**, while the frequency range that is stopped by the filter is called the **stop band**. These bands can be as large or as small as desired and can be used to design systems that support everyday life.

Passive filters are defined by their ability to filter out signals using only passive elements. They do not require an external power source and can complete their intended function using only the input signal. Most passive filters are linear and are constructed using basic linear elements – resistors, capacitors, and inductors. The fact that we distinguish passive filters from other types of filters would seem to imply that there are active filters. There are, and you will learn about those in a later course.
Passive filters have a few advantages over active filters:

- Guaranteed stability
- Scale better with larger signals
- Inexpensive (typically)

We have introduced quite a few new topics in this experiment, but these are all foundational to understanding how filters work. In essence, a passive filter is just a frequency dependent voltage divider. There is nothing special going on when you combine these components; the special part comes from how you combined them. As you will see, because of the impedance of the inductors and capacitors, you can control which signal frequencies exit your filter and which are filtered out.

7 Low-Pass Filter

Low-Pass filters do exactly what their name implies. Low frequencies can pass through the filter while higher frequencies are rejected. This is very useful when you are trying to extract base-band signal (signal very near zero compared to the highest frequency.) This is how an AM (amplitude modulated) signal works. The carrier frequency is much higher than the audio frequency of the base-band signal. AM radios in the United State range from 540kHz to 1700kHz. This allows the radio to filter out the carrier signal while maintaining the audio signal.
Now let's look at how we can construct a low pass filter.

7.1 RC Low-Pass Filter

First, let's look a the structure of a low pass filter using a resistor and capacitor. As you can see, the resistor and

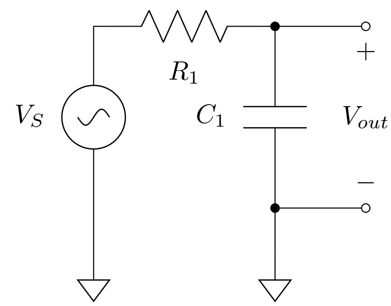


Figure 4: R_2 replaced by C_1 in a voltage divider creating a low-pass filter

capacitor create a voltage divider just like we saw in Experiment 2, but instead of R_2, we see C_1. This change means that the voltage divider is now frequency dependent. As you should remember, a capacitor acts as an open circuit at DC (or low frequencies) and as a short at ∞ frequency (or high frequencies.) So if we think about what is happening at lower frequencies, there is no direct path to ground, so all the current will follow the path of least resistance through V_{out}. We will see as the frequency increases that the capacitor starts to act like a short, and the path of least resistance is through the capacitor to ground. This means that less power will be delivered to V_{out}. If you plot V_{out} vs. frequency, you should get a graph that looks similar to this:

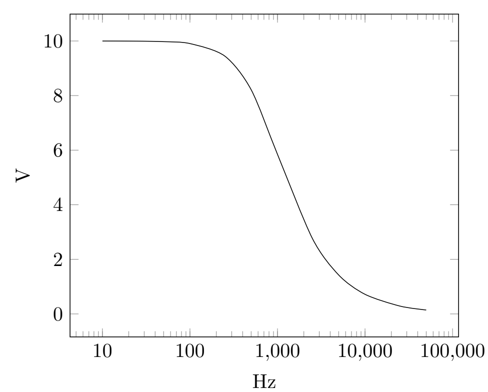


Figure 5: Frequency Response of a Low-Pass Filter

Where the -3dB (half-power) is at the cutoff frequency of your filter.

Calculating the cutoff frequency of a first order filter (a filter with that is described mathematical using a first order differential equation) is fairly straightforward. Here we will walk you through the derivation, but you will only need to focus on the final equation. We have defined our cutoff frequency to be the -3dB point. At that point our gain V_{out}/V_{in} should be $1/\sqrt{2}$. Taking that knowledge and the knowledge we have about voltage dividers, we get:

$$V_{out} = V_{in} * \frac{-jX_c}{R_1 - jX_c} \tag{23}$$

Rearranging the equation we get:

$$\frac{V_{out}}{V_{in}} = \frac{-jX_c}{R_1 - jX_c} \tag{24}$$

Replace your gain with $1/\sqrt{2}$:

$$\frac{1}{\sqrt{2}} = \frac{-jX_c}{R_1 - jX_c} \tag{25}$$

Then we take the magnitude of the complex expression:

$$\begin{aligned} \left|\frac{-jX_c}{R_1 - jX_c}\right| &= \frac{1}{\sqrt{2}} \\ \frac{X_c}{\sqrt{R_1^2 + X_c^2}} &= \frac{1}{\sqrt{2}} \\ \frac{1}{\sqrt{(R_1^2/X_c^2) + 1}} &= \frac{1}{\sqrt{2}} \\ (R_1^2/X_c^2) &= 1 \\ R_1 &= X_c \end{aligned} \tag{26}$$

As you can see, the resistance of the resistor and the reactance of the capacitor should be equal. Now, if we replace the capacitor reactance with Equation 1, we will get:

$$\begin{aligned} R_1 &= \frac{1}{C_1\omega} \\ \omega &= \frac{1}{R_1C_1} \\ f_c &= \frac{1}{2\pi R_1C_1} \end{aligned} \tag{27}$$

7.2 RL Low-Pass Filter

Now let's look a the structure of a low pass filter using a resistor and inductor.

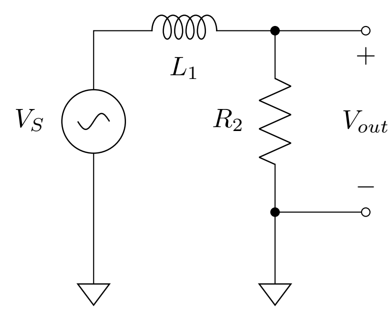


Figure 6: R_1 replaced by L_1 in a voltage divider creating a low-pass filter

Very similar to the RC filter, the RL filter replaces one of the resistors in the traditional voltage divider with an inductor. This allows this voltage divider to be frequency dependent as well, the difference being that the frequency response of the inductor will short at low frequencies and cause an open circuit at high frequencies. We will get a similar frequency response to the RC Low-pass and, as such, the graphs should look almost identical. The derivation is also very similar, but we replace X_c with X_l. So our cutoff frequency (f_c) will be:

$$f_c = \frac{R_2}{2\pi L_1} \tag{28}$$

8 High-Pass Filter

In many ways, a high pass filter is the opposite of a low pass filter. With a high pass filter, the high frequencies pass through while the low frequencies are blocked. This is useful to block DC from a circuit that should not see it (think AC coupling from the oscilloscope experiment). It is also useful while mixing audio signals (keep this in mind for the final project in this course.) The frequency response of a high pass filter should look like this:

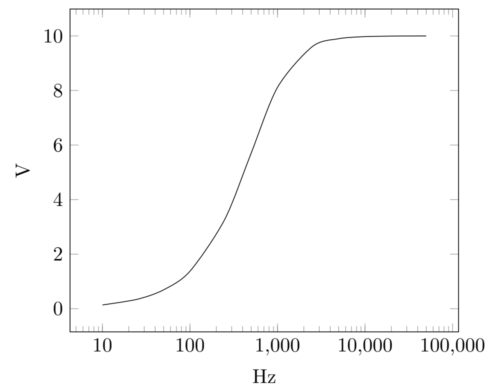


Figure 7: Frequency Response of a High Pass Filters

Because the physical principles are the same, you should be able to logically understand how each of these circuits work.

8.1 RC High-Pass Filter

Looking at the structure of an RC High-Pass filter, it is still just a frequency dependent voltage divider, instead now we are replacing R_1 with C_1.

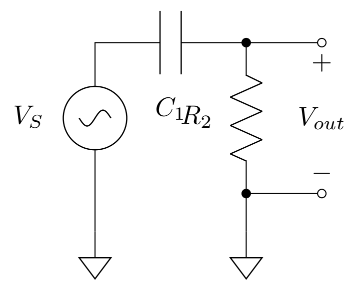


Figure 8: R_1 replaced by C_1 in a voltage divider creating a high-pass filter

It turns out that running though the calculations are exactly the same. We won't go through the full derivation, but:

$$\frac{V_{out}}{V_{in}} = \frac{R_2}{R_2 - jX_c} \tag{29}$$

Doing the same derivation we end up at:

$$f_c = \frac{1}{2\pi R_2 C_1} \tag{30}$$

As you can see, this is the same equation for the low pass filter. Because the impedance of both elements needs to be equal, the equation works out to be the same if you are looking at a low pass or high pass filter. (**Remember this only holds for the -3dB point**.)

8.2 RL High-Pass Filter

Again, the principles are the same here. The circuit shorts to ground for the low frequencies and continues to the load for high frequencies.
The general voltage divider equation still holds and it comes out to this:

$$\frac{V_{out}}{V_{in}} = \frac{R_2}{R_2 + jX_l} \tag{31}$$

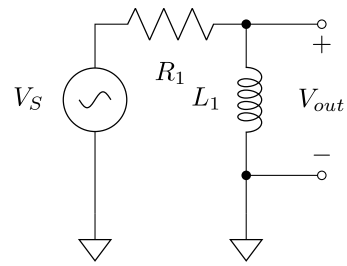


Figure 9: R_2 replaced by L_1 in a voltage divider creating a high-pass filter

Doing the same derivation we end up at:

$$f_c = \frac{R}{2\pi L} \tag{32}$$

9 Band-Pass Filter

There are multiple ways to tackle a band pass filter. Using the knowledge you now have about LPFs and HPFs, we can see that if you place a low pass filter in series with a high pass filter, you should get a band pass filter. This is assuming your cutoff frequencies are sufficiently selected (if not you may make a notch or band stop filter). While fundamentally this concept is pretty easy to follow, in practice there is much more to take into account. Because the two circuits in series introduce loading and will affect the over all impedance of both filters, we need to take that into account while calculating the cutoff frequencies.

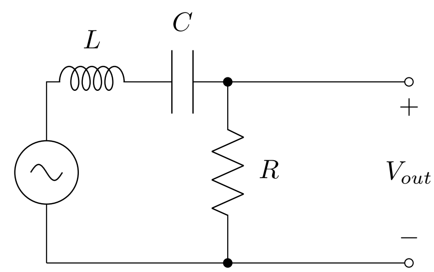


Figure 10: RLC Band Pass Filter

Looking at Figure 10, we can see for an RLC band pass circuit the inductor and the capacitor are in series with each other. This means that the impedance of the two components combined would be:

$$\begin{aligned} Z_{LC} &= Z_L + Z_C = j\omega L + \frac{1}{j\omega C} \\ &= j\left(\omega L - \frac{1}{\omega C}\right) \end{aligned} \tag{33}$$

For a band pass filter we are concerned about two aspects, the center frequency and the bandwidth. The center frequency, or resonant frequency, is the frequency that is at the center of the band and represents when the impedances of the reactive components sum to 0, leaving a purely resistive circuit. The center frequency can be calculated using the equation:

$$\omega_o = \frac{1}{\sqrt{LC}}, \quad f_c = \frac{1}{2\pi\sqrt{LC}} \tag{34}$$

9.1 Band Width

The band width is the frequency range where the filter is above your selected cut-off frequency (-3dB in this lab), and is controlled by the cutoff frequencies of the low pass and high pass filters that control the pass band of the band pass filter. These can be calculated similarly to how the cutoff frequencies are calculated for either filter, but you need to take all impedances into account.

In later experiments you will learn about buffers that can be used to isolate filters so they do not affect each other. This allows for cleaner calculations, but does introduce circuit complexity and more noise into the system.

9.2 Band Pass Filter Band Width

To be able to calculate the band width of a band filter, it is helpful to be able to calculate the **transfer function**. The transfer function is another term for frequency response. It is the ratio of voltage out over voltage in. Given that information, we should be able to derive the transfer function of the above circuit:

$$
\begin{aligned}
V_{out} &= V_{in} * \frac{Z_{LC}}{R + Z_{LC}} \\
\frac{V_{out}}{V_{in}} &= \frac{ZLC}{R + Z_{LC}}
\end{aligned}
\tag{35}
$$

Plugging Z_{LC} into equation 35 we end up with:

$$
\begin{aligned}
\frac{V_{out}}{V_{in}} &= \frac{j\left(\omega L - \frac{1}{\omega C}\right)}{R + j\left(\omega L - \frac{1}{\omega C}\right)} \\
\frac{V_{out}}{V_{in}} &= \frac{jR\omega}{-L\omega^2 + jR\omega + \frac{1}{C}}
\end{aligned}
\tag{36}
$$

Now that we have the transfer function, we can set its magnitude equal to half power $\left(\frac{1}{\sqrt{2}}\right)$ and solve for ω. Because of the quadratic in the denominator, we will have two valid solutions for this equation. This quadratic is not a fun one to solve, mainly due to j being in the equation, and the quadratic is on the bottom of the fraction. You will learn tricks in solving equations like these in later courses. These two solutions will be your high and low cutoff frequencies.

$$
\begin{aligned}
\omega_{c1} = 2\pi f_{c1} &= -\frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 + \frac{1}{LC}} \\
\omega_{c2} = 2\pi f_{c2} &= \frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 + \frac{1}{LC}}
\end{aligned}
\tag{37}
$$

The bandwidth of the filter is then $\omega_{c2} - \omega_{c1}$, which we will refer to as β. By doing some quick algebra, we can see that:

$$
\beta = \frac{R}{L} \tag{38}
$$

We know that "quick algebra" is normally not "quick", but because we are taking the difference of the two equations, the radicals cancel out you are left with $\frac{R}{2L}$ from each equation. Please remember that this derivation is for the series LC bandpass filter and you will need to go through the whole process again for other types of 2nd order filters. For example, the bandwidth for a parallel RLC circuit (the inductor and capacitor are in parallel with each other) is $\beta = 1/RC$.

9.3 Qualtity Factor

The qualtity factor (or Q) of a filter is the ratio between the center frequency and the bandwidth.

$$
Q = \frac{\omega_o}{\beta} \tag{39}
$$

This ratio measures the "sharpness" of the peak of the fitler. The higher that Q is, the more narrow the bandwidth is. While this indicates that the bandwidth will allow frequencies at or near the center frequency through, that does not mean it is what you as the engineer want. Sometimes a low Q is desirable for the overall function of the system.

10 Notes on Passive Filters

Filters are extremely helpful circuits. Filters are relatively simple to setup, and once you get used to the math, figuring out their characteristics is fairly straightforward. That said, just like literally everything else in electronics, filters, especially passive filters, have their limitations.

Remember that a passive filter is a fancy voltage divider that is dependent on the frequency applied in the circuit. Think back to the voltage divider lab, and recall that we cannot step up voltages in a voltage divider, we can only step them down. That means we will never get a gain higher than 0 with a lone passive filter. We can amplify the signal after it goes through the filter (which we will see next lab), but the filter isn't the part doing the amplification.

It is sometimes worth thinking about your high and low pass filters with a bandwidth. You just have to determine yourself if it makes sense to think of it as a stop or a pass band. With a low-pass filter, the pass band goes from 0Hz to the cutoff frequency, and the stop-band is from the cutoff frequency to whatever maximum frequency makes sense for your circuit. The opposite is true for your high-pass filter, with your pass and stop bands flipped. The perspective you take on the filter will depend on your application, so it might be worth thinking of the filter as a pass-band and a stop-band simultaneously, and determining which perspective works best at that time.

Pre-Lab

11 Low-pass Filter

1. What is the pass band of a low pass filter constructed from a 1.6kΩ resistor and a 10nF capacitor?
2. Given a cutoff frequency of 5kHz and a known capacitance of 470nF, what would be the required resistor value be to acheive the desired cutoff frequency?
3. Given the component values from your lab kit, draw the schematic for an RL LPF with a cutoff frequency of about 29kHz. To draw a schematic, please use Spice or another schematic software.
4. Using a circuit simulation software (Spice), complete a frequency analysis of the circuit in Question 3. **Capture** the frequency analysis plot.
   HINT: Use a signal component for your independent voltage source. Set functions to "none" and AC amplitude to 1.
   HINT: The use the .ac simulation command if you are using LTspice, or "ac sweep" in Multisim.

12 High-pass Filter

1. What is the stop band of a high pass filter constructed from a 1.6kΩ resistor and a 10nF capacitor?
2. Using component values from your kit, construct an RC High Pass Filter in Spice with a cutoff frequency such that a 100kHz sine wave will pass through, and a 1kHz sine wave will be stopped.

13 Variable Band Filter

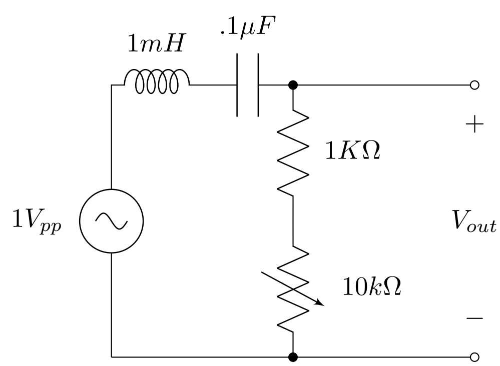


Figure 11: RLC Band Pass Filter

1. For the band filter above, calculate the center frequency of the band pass filter in Figure 11.
   Do you need the value of the potentiometer to calculate the center frequency?
2. Calculate the band width of the band pass filter when the 10kΩ variable resistor is 0Ω.
3. Calculate the band width of the band pass filter when the 10kΩ variable resistor is 10kΩ.
4. Simulate the filter in LTspice using frequency analysis, then verify your center frequency and band width calculations.

In-Lab

14 Manual Frequency Response Analysis

1. Construct the RL circuit from the Prelab on your breadboard (RL Filter).
2. Set your function generator to a 0.5V amplitude sine wave (1Vpp).
3. Connect the function generator to the input of the circuit and measure the output with the scope.
4. Sweep (manually adjust) the input frequency from 10Hz to 100kHz, taking amplitude magnitude measurements for at least 5 points per decade.
5. Record your V_{in} and V_{out}.
6. Plot V_{out}/V_{in} (gain) vs. frequency using MATLAB, numPy, or Excel.

15 Exploring FRA on the Oscilloscope

1. Continue using your constructed circuit from the Prelab(RL Filter).
2. Connect the "Gen Out" of the scope to the input of your circuit.
3. Connect Channel 1 to your input as well and Channel 2 to your output of your filter.
   Make sure ground is common for all connections
4. Using the Analyze function of the scope find the Frequency Response Analysis (FRA) tool.
5. Set the frequency parameters to show both the stop band and the pass band of the filter (you want to be able to see both high and low gain on your filter).
6. Run the frequency response tool.
7. Collect a screenshot of the Gain vs. Frequency Graph.
8. Collect a .csv file of the FRA data.
9. Using the .csv, file plot the Gain vs. Frequency Graph in MATLAB, numPy, or Excel.
10. Compare the cutoff frequency calculated in the prelab with the cutoff frequency of the found using the FRA tool.
11. What parts of the system could introduce error, and how should we mitigate the error?
12. repeat the above steps with the RC High-pass filter calculated in the prelab.

16 Variable Band Filter

1. Construct the band filter you calculated and simulated in the prelab.
2. Collect the FRA plots for the extremes of the system from 10Hz to 1MHz. Verify your center frequency and band width and determine the amount of error in your system.
   Note: You may need to switch your probes to 10x mode to get a proper reading. Why might this be?
3. Is this a band pass or a band-reject filter? Where can this be used?
4. What does varying the variable resistor do to the system? How is this helpful? Where could this be used?
5. Repeat the above steps, except read across the *reactive* components of the circuit. Make sure you are measuring across both components! You will likely have to rearrange your circuit so the reactive components have a direct connection to the common ground.