Lab 5: Wave Behavior and Filter Design

Prelab

When dealing with DC power sources things stay fairly simple, but once you have a power source with an oscillating voltage all sorts of complications arise. One of these has to do with how waves behave at the boundaries of materials of unequal impedance. Many helpful analogs can be drawn between wave behavior in electronics and wave behavior in mechanics, optics and acoustics.

Watch the following video by Bell Labs (30 min) and write a short paragraph describing what impedance is, why it needs to be “matched”, and how one would go about performing such matching in various mediums.

https://www.youtube.com/watch?v=DovunOxlY1k

You will be referencing sections 2.35.1 (Fourier Series) and 9.1-9.7 (Passive Filter Design) during lab. Don’t worry about reading those sections in depth before lab, but do take a few minutes to skim though and get a feel for what they cover.

Be sure that you or your partner bring the textbook and a laptop to lab.

Part I: Band Pass Filters and Fourier Series

Last week you built a band pass filters using a resistor, capacitor, and an inductor. You’ll be examining the same circuit this week but instead of using a sine wave as the input, you will use a square wave.

1.1 Fourier Series

Fourier’s theorem states that any reasonably periodic signal can be decomposed into an infinite sum of sine and/or cosine terms with various amplitudes.

In section 2.35.1, your book shows how a square wave can be represented as an infinite sum of sine waves. Figure 2.215 on page 239 shows the first three terms of said sum, and that the result of summing just those first three already evokes the appearance of a square wave.

- Write down the first FOUR terms in the V(t) sum for the Fourier series decomposition of a square wave.
- In terms of ω_0, what will the frequencies be for the sine wave in each of these terms?

1.2 Filter response in the time domain

The band pass filter nearly perfectly lets through a narrow range of frequencies while heavily suppressing the rest. Using a sine wave we are only putting in a single frequency at any given time. However, as noted above, a square wave is composed of many frequency components.

Take a picture of each output for reference, but you only need include the first one in your report.

1. Measure the components for use in the band pass filter and calculate the central frequency using $f_c = 1/(2\pi\sqrt{LC})$.
2. Reset the scope using DEFAULT SETUP. Be sure both the probes and channels are in 10X mode and AC coupled. Leave averaging turned off for now.
3. Construct the band pass circuit shown in figure 1 and set the function generator to a square wave with a frequency equal to the one you just found. Overlay the input and output signals on the scope screen.
4. *Explain what you see in terms of Fourier components.*
5. Reduce the frequency of the square wave and watch what happens. Specifically examine the output at f_c/n where $n = 2, 3, 4, 5, 6, 7$. *Which of these correspond to Fourier components? What do you notice at these frequencies?*

The filter only allows a narrow range around f_c pass through unattenuated. When you input a square wave with a primary frequency of $f_c/3$, then f_c is the frequency of the $n = 3$ Fourier component. The filter will attenuate all frequencies except this $n = 3$ component. A similar behavior will occur at each odd integer division of f_c.

How does the output waveform at $f_c/3$ compare to the $n = 3$ Fourier component shown in your textbook?

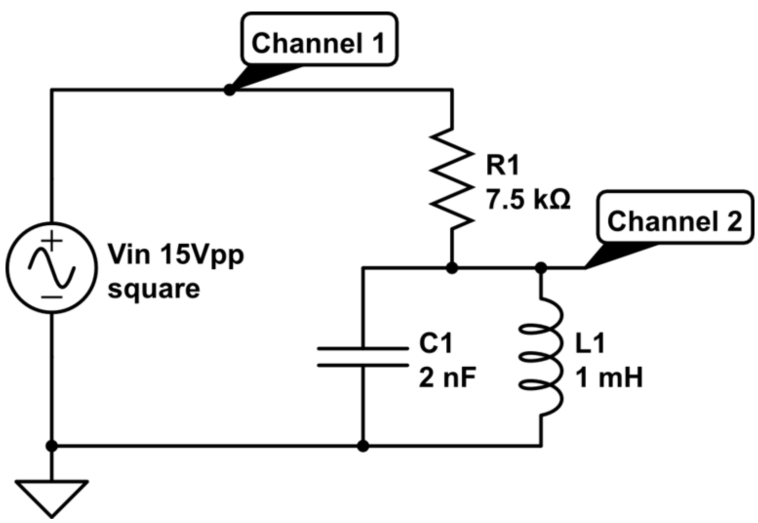

Figure 1: Band Pass Filter

1.3 FFT

Your oscilloscope has the ability to perform a real time Fourier transform of the signals it measures. (Called an FFT or Fast Fourier Transform.)

1. Set the frequency back to f_c.
2. Go to the MATH MENU and change the operation to FFT. Change the window to "Hanning" and use the horizontal zoom knob to change the horizontal scale to 125 kHz per division. I also suggest keeping averaging off.
3. With the source as channel 1 you should see a fairly noisy signal, but with a few narrow peaks that stand out above the rest. *Estimating by eye, how do the locations of these peaks compare to the frequencies you found in section 1.1 for the first few Fourier components?*
4. Switch to channel 2. *What happens? Include a picture in your report.*
5. Slowly reduce the frequency and watch as the primary component moves to the left and the amplitude gets reduced. The $n = 3$ component should become accentuated when it equals the bandpass frequency, f_c. *Does this happen?*

1.4 Frequency Sweep

Now you will set up the frequency sweep just as you did last week and compare result of inputing a sine wave vs a square wave.

1. Press the MATH MENU button to exit the FFT mode.
2. Refer back to lab 4 if you need help remembering how to set up the frequency sweep.
3. What is different about the output when a square wave is used as the input as opposed to a sine wave? Include a picture in your report.

Part II: Multistage RC Filters

Design a two stage low pass filter with a cut off frequency of approximately 16 kHz by chaining together two single stage low pass filters as shown in figure 2.

- The two stages should each have a cutoff frequency of 16 kHz
- R_2 should be an order of magnitude (or more) larger than R_1 to avoid unduly loading the first stage
- Don't use electrolytic capacitors
- Avoid very low resistor values (under 25Ω) and very high values (over $100\ \text{k}\Omega$).

The filters you built previously were designed such that the output impedance of the function generator had very little effect on the measurements. Depending on what values you choose, that may or may not hold true this time. Be sure you record the amplitude of the input wave form (V_1) and output waveform (V_{out} at each frequency.

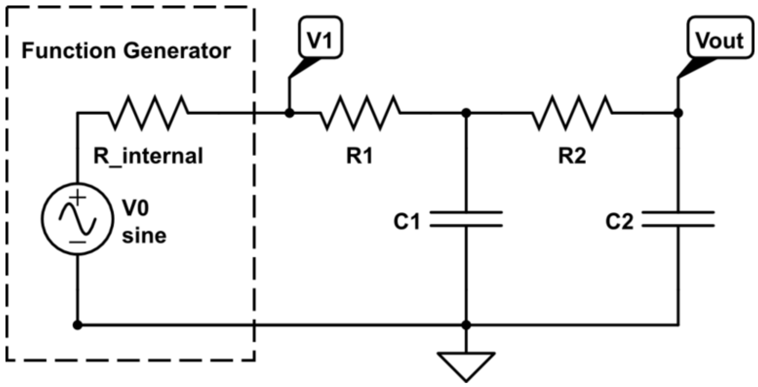

Figure 2: Two stage low pass filter

1. Reset the scope using DEFAULT SETUP. Set both channels to AC coupling and 10X probe mode (of course checking that the probes are in the proper mode as well). Using an external trigger can make things a bit easier but is not necessary.
2. Measure the values of the resistors and capacitors you chose and record them in your lab report as R1, C1, R2, and C2.
3. Calculate and record the cutoff frequency for each stage (f_1 and f_2) using these measured values.
4. Connect the function generator directly to the scope and set the amplitude to 20 volts peak-to-peak. This is V_0 in the figure, but you can't actually measure that value because there is always some voltage drop across the internal resistance. V_0 and V_1 will be nearly identical when very little current is being drawn.
5. Build the circuit connecting channel one to measure V1 and channel 2 to measure V$_{out}$
6. For frequencies starting at 100 Hz and doubling up to 204.8 kHz, record V_1 and V$_{out}$. Include a data point at the average of f_1 and f_2. V_1 may stay nearly the same or may vary quite a bit depending on the component values you chose. Either is fine.
7. Calculate the decibels of attenuation for each data point using $[dB] = 20\log(V_{out}/V_1)$.
8. Calculate the difference in decibels between each step. As you get far from the cutoff frequency, what does this value become? It should be approximately twice the dB/octave slope from last week.
9. What attenuation do you find for the cutoff frequency? How does that compare to what you found for a single stage filter last week?
10. Create a Bode plot of the results just as you did last week.

Part III: Butterworth Filter Design

3.1 Third Order

Following the instructions in chapter 9 of your textbook, and the scanned pages of the Art of Electronics which are available on Moodle, design and build a third order Butterworth low pass filter with a 50Ω load and a cutoff frequency of 16 kHz.

NOTE THAT IN FIGURES 9.5 and 9.7 IN YOUR TEXTBOOK THE COMPONENT LABELED C_2 SHOULD BE C_3. THE DIAGRAMS IN THE ART OF ELECTRONICS ARE CORRECT.

Take data points from 100 Hz and then doubling up to 102.4 kHz and add the resulting data to the same Bode plot.

Include your calculations and a circuit diagram with labeled values in your report.

What does the slope approach in terms of dB/octave? How does this compare to that of the RC filter?

What are the advantages and disadvantages of Butterworth filters as compared to other filter types (such as Bessel, Chebyshev, etc...)?

3.2 Fifth Order

With any time you have left, try to design a fifth order Butterworth filter with a 50Ω load and a cutoff frequency of 13 kHz.

If you don't finish this section you won't loose any points. If you get it working correctly and create the Bode plot, I'll give you 5 points of extra credit.