

# 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.

The video is a bit slow, I suggest watching it at 1.25x or 1.5x speed.

<https://www.youtube.com/watch?v=Dovun0x1Y1k>

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.

If you want to have enough time for the extra credit at the end, you should do some of the circuit design calculations before lab.

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

## Supplies

- Multimeter
- Function Generator (x2)
- Oscilloscope (and probes)
- Breadboard
- Minigrabber probes w/ adapters
- 7.5 k $\Omega$  resistor
- 1nF capacitors
- 1000 $\mu$ H inductor
- Assorted resistors, capacitors and inductors to be determined during 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.

- Q. Write down the first four terms in the  $V(t)$  sum for the Fourier series decomposition of a square wave.
- Q. In terms of  $f_0$ , what will the frequencies be for the sine wave in each of these terms? (Don't over think it, recall  $\omega_0 = 2\pi f_0$ )

## 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.**

- Measure the components for use in the band pass filter and calculate the central frequency using  $f_0 = 1/(2\pi\sqrt{LC})$ .
  - 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.
  - 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. Include a picture in your report.
- Q. Explain what you see in terms of Fourier components.
- Reduce the frequency of the square wave and watch what happens. Specifically examine the output at  $f_0/n$  where  $n = 2, 3, 4, 5, 6, 7$ .

- Q. Describe how the output differs between when  $n$  is even and when  $n$  is odd.

The filter only allows a narrow range around  $f_0$  to pass through unattenuated. When you input a square wave with a primary frequency of  $f_0/3$ , then the frequency of the  $n = 3$  Fourier component equals the pass frequency. The filter will attenuate all frequencies except this  $n = 3$  component.

For example, say the band-pass frequency were 6 kHz and you input a square wave with a frequency of  $6/3 = 2$  kHz. The  $n = 3$  harmonic would be  $3 * 2 = 6$  kHz, which, being equal to the band-pass frequency, would be let through the filter with nearly zero attenuation. A similar behavior will occur at each odd integer division of  $f_0$ .

- Q. Compare the output waveform when at a frequency of  $f_0/3$  to the  $n = 3$  Fourier component shown in your textbook.

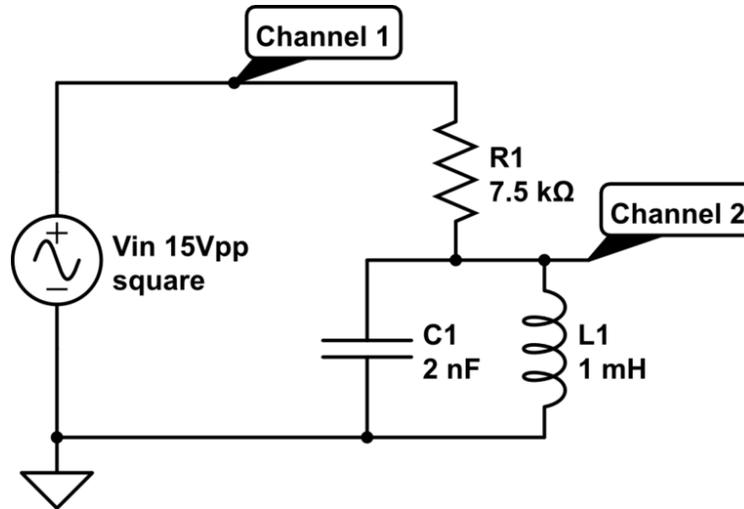


Figure 1: Band Pass Filter

### 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.)

- Set the frequency of the square wave back to your calculated  $f_0$ .
  - 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. Keep averaging off.
  - With the source set to channel 1, you should see a fairly noisy signal with a few narrow peaks that stand out above the rest.
- Q. Estimating by eye, write down the locations (frequencies) of the first several peaks. Compare the locations of these peaks to the frequencies you wrote down in section 1.1 for the first few Fourier components.
- Switch the source to channel 2 so that the input to the FFT is the output of your filter circuit. You might have to zoom back to 125 kHz/div
- Q. Describe the difference between the FFT of channel 1 and of channel 2. Why does this difference exist? Include a picture in your report.
- 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,  $f_0$ .
- Q. Take a picture of the FFT when the  $n = 3$  component of the square wave is equal to the bandpass frequency. Include this picture in your report.

## 1.4 Frequency Sweep

Now you will set one function generator with a triangle wave and use it to continuously sweep the frequency of the other function generator, just as you did last week.

- Press the Channel 1 button to exit the FFT mode.
  - Refer back to the end of lab 4 if you need help remembering how to set up the frequency sweep.
- Q. What is different about the output when a square wave is used as the input as opposed to a sine wave? Include a picture of each 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 ( $f_c = 1/(2\pi RC)$ )
- $R_2$  should be at least an order of magnitude larger than  $R_1$  to avoid unduly loading the first stage
- Don't use electrolytic capacitors
- Avoid low resistor values (under  $500\Omega$ ) and very high values (over  $100\text{ k}\Omega$ ).

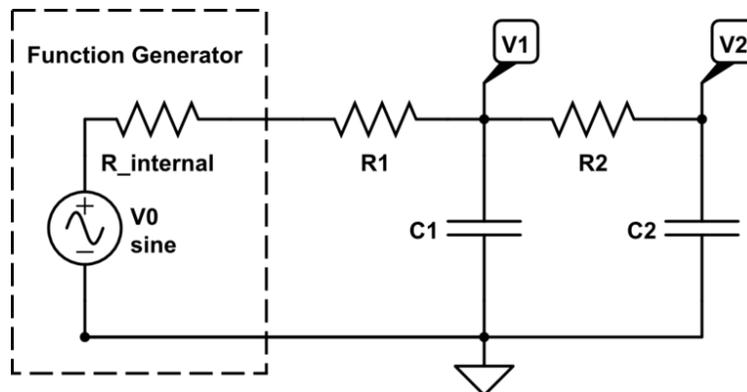


Figure 2: Two stage low pass filter

- 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). Check that the probes are properly compensated. Using an external trigger can make things a bit easier but is not necessary.
- Q. Measure the values of the resistors and capacitors you chose and record them in your lab report as  $R_1$ ,  $C_1$ ,  $R_2$ , and  $C_2$ .

- Q. Calculate and record the cutoff frequency for each stage ( $f_{c1}$  and  $f_{c2}$ ) using these measured values. **Include a full uncertainty propagation.**
- Build the circuit. Connect channel 1 to measure  $V_1$  (the output of the first stage) and channel 2 to measure  $V_2$  (the output of the second stage).
  - The output voltage attenuation will be calculated relative to the DC gain. Set the frequency to around 50 Hz (low enough to approximate DC) and record  $V_1$  and  $V_2$ . Label these as  $V_{1DC}$  and  $V_{2DC}$ .
  - For frequencies starting at 100 Hz and doubling up to 204.8 kHz, record  $V_1$  and  $V_2$ .
  - Calculate the decibels of attenuation for each data point using  $[dB] = 20 \log(V_{out}/V_{DC})$ .
- Q. Calculate the difference in decibels between each step. As you get far from the cutoff frequency, what does this value approach for each stage? (The second stage should be approximately twice the dB/octave slope of the first stage.)
- Q. Create a Bode plot of the results showing the attenuation after each stage.

## Part III: Butterworth Filter Design

### 3.1 Third Order

Following the instructions in section 9.3 of your textbook (starting on Step 4), 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  $50\Omega$  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 have been labeled  $C_3$ . The diagrams in The Art of Electronics are correct.**

- Measure the output voltage at 50 Hz and use this value as  $V_{DC}$  just as in the previous section.
  - Take data points starting at 100 Hz and then doubling up to 102.4 kHz.
- Q. Add the resulting data to the same Bode plot you made in the previous section.
- Q. Include your calculations and a circuit diagram with labeled values in your report.
- Q. What does the slope approach in terms of dB/octave? How does this compare to that of the RC filter?
- Q. What are the advantages and disadvantages of Butterworth filters as compared to other filter types (such as Bessel, Chebyshev, etc...)?

You should have a single Bode plot in your report with three curves. One curve for the first stage of the RC filter, one for the second stage, and one for the Butterworth filter.

### 3.2 Fifth Order - Extra Credit [5 pts]

Design a fifth order Butterworth filter with a  $50\Omega$  load and a cutoff frequency of 13 kHz. Include a schematic and your calculations in the lab report. Add the resulting data to the same Bode plot. If you complete this section, the Bode plot should have 4 curves.