Lab 4: Active Band-Pass Filter Project

Electronics II

Contents

Schedule

Day 1: Present your design calculations for your filter. Verify your design through simulation.

Day 2: Demonstrate your working filter to the TAs and take measurements for your report.

In order to be able to complete this lab in two weeks, students must come into the first lab with at least the paper design of the filter completed and be ready to start simulation.

Purpose

The purpose of this laboratory is to construct a filter from standardized filter blocks. Students will study second-order filter circuits and the Chebyshev filter response in the context of a design problem.

Note that modern filter design does not use the exact circuits used in this lab. However, the mathematical process used in this lab is still valid, and the design process used for modern digital filters or transmission line filters is still very similar.

Introduction

Before beginning this project read all the lecture notes pertaining to filters, and the data sheet for the TL082CD op-amp which you will be using in this lab. Also read Appendix B, found on cuLearn for additional information.

Chebyshev Filter Design Requirements

Humans can hear sounds that range in frequency from about 20 Hz to 20 kHz (although many of us can't hear quite that low or high). However, many types of speakers cannot reproduce the full range of frequencies well. To make up for this, good audio speakers for sound systems often contain three or more cones of different sizes optimized to produce sound in a range of frequencies. To ensure an even response for the entire speaker system, the inputs of each cone should be filtered. In this lab you will be designing a bandpass filter to drive a mid-range cone.

You will need to design, build, and test/demonstrate an active filter with the following specifications:

1. Band-pass action
2. Chebyshev response
3. 3 dB passband ripple
4. Fourth-order roll-off (**think hard about this or you will make a mistake that will make your life hard!**)
5. Lower cut off frequency and upper cut-off frequency as calculated below.
6. Target passband gain = +0.0 dB to -3.0 dB
7. ±8% error allowed in $f_{-3\ \mathrm{dB}}$
8. ±1.0 dB error allowed in passband gain
9. Supply voltages to be ±15 volts
10. Op-Amps to be type **TL082CD** (No more than 6 Op-Amps total)
11. Output voltage swing > ±10 volts

Calculating your lower and upper cutoff frequencies:

- Start with X, which is your student number.
- Calculate **A = X mod 1031 and B = X mod 1033** (note mod is the modulo operation)
- Calculate $f_{-3\,dB\,Lower} = \frac{A^5}{5.534x10^9} - \frac{A^4}{2.11x10^6} + \frac{A^3}{2287} - \frac{A^2}{6.1} + 20.2A + 750$

- Calculate $\delta = -\frac{B^2}{180\,000} + \frac{B}{173} + 0.5$
- Calculate $f_{-3\,dB\,Upper} = f_{-3\,dB\,Lower}(1+\delta)$

To implement the filter, you will construct and combine two second order filters. Students will need to decide which circuits to use and what order to put them in. Since you have 6 op-amps to work with, using the Tow-Thomas (with Feed-forward) and/or KHN circuits are recommended.

For each of the two individual second-order sections, as well as the overall Chebyshev filter, make accurate measurements of the gain over the range of frequencies covering at least 0.1*$f_{\text{-3 dB Lower}}$ to 10*$f_{\text{-3 dB Upper}}$.

Finally, determine the input impedance of the filter at very low and very high frequencies and compare them to theoretical values.

Some Further Information

1. It is not necessary to correct output inversion, if any is present, in the pass-band.
2. Try to keep all resistor values greater than or equal to 3.3 kΩ so as not to exceed ±10 mA output current in the op amps by too much, and also to maintain the maximum peak output voltage.
3. Try to keep most resistor values below 100 kΩ to keep capacitor values high, in order to minimize the effects of stray circuit capacitances, and input bias currents. The only exception is R_q of the Tow-Thomas Biquad, which doesn't matter because of the existence of R_3 and R_g.
4. All resistor values must be kept to the following values in powers of 10 (i.e.: 180, 1.8k, 18k, 180k): 1, 1.2, 1.5, 1.8, 2.2, 2.7, 3.3, 3.9, 4.7, 5.6, 6.8, 8.2
5. For accuracy, mix components as required to get desired component values.
6. Available capacitors for your design: (2 of each) 3.3 nF, 6.8 nF, 10 nF, 15 nF, 22 nF, 33 nF, 47 nF, 68 nF, 0.1 μF. Keep this in mind when you make your design choices.

Design Approach

Like the amplifier design, there is no unique solution to the filter design problem that has been laid out because of the fact that there are more variables in the circuit than there are design parameters that have to be met. Consequently, as in the amplifier project, a design solution requires the judicious fixing of some of the unspecified degrees of freedom in the circuit in order to begin to solve the problem. In the next sections are presented the highlights of some example filter design techniques.

Example of Scaling Transfer Functions

As detailed in your notes (page B18), standard transfer function forms for filters are all normalized to a cut-off frequency of 1 rad/s. Of course, it is very rare to build a filter at this frequency, but transfer functions can easily be converted to any cutoff frequency we need, simply by using a variable substitution. 1 rad/s is chosen since it makes this process easier.

Suppose we have a part of a transfer function which we are told is normalized to 1 rad/s.

$$T(s) = \frac{0.25}{s^2 + s + 0.25}$$

Plotting the magnitude response gives us Figure 1.

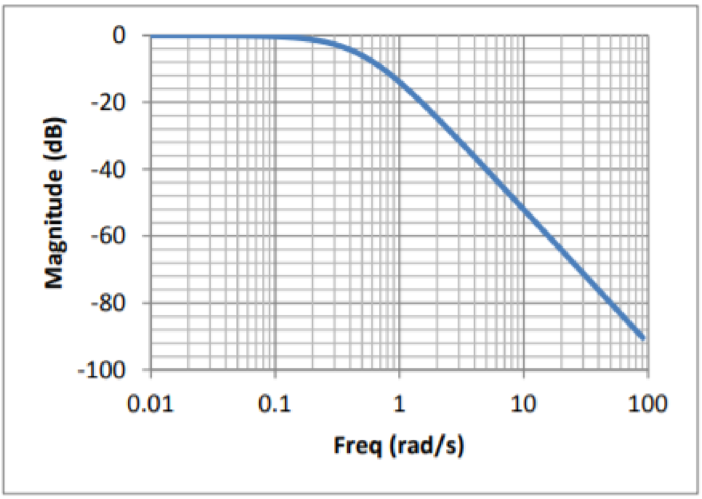

Figure 1: Magnitude Response of a Transfer Function

Note that at 1 rad/s, the magnitude is not -3 dB. Just because the transfer function is normalized to 1 rad/s does not mean that that is the corner frequency for all parts. The entire equation for a normalized filter (which may include numerous transfer functions) is calculated so that the 3 dB corner frequency of the entire filter is at 1 rad/s. However, the equation itself may consist of many sub-functions, each of which will have their own corner frequency. In addition, for second order rational equations, the 3 dB frequency is usually not the same as the natural frequency.

The transfer function shown before has $\omega_o = \sqrt{0.25} = 0.5$ rad/sec, $\frac{\omega_o}{Q} = 1$, thus $Q = \omega_o = 0.5$ and $H_o = 1$.

If we want to scale this transfer function to 1000 rad/s, we will replace s with s'/1000 to get

$$T(s') = \frac{0.25}{(\frac{s'}{1000})^2 + (\frac{s'}{1000}) + 0.25} = \frac{250\,000}{s'^2 + 1000s' + 250\,000}$$

now $\omega_o = \sqrt{(1000)^2 * 0.25} = 1000$ x $0.5 = 500$, $Q = \frac{\omega_o}{1000} = 0.5$, $H_o = 1$

Thus with scaling, Q and H_o remain the same and only ω_o is changed. Scaling can also be done to convert a band-pass filter response into high-pass, band-pass, band-stop, etc. filter blocks.

Example Design Approach for a Low-Pass Butterworth Filter

As an example, a Butterworth response filter is designed to meet the requirements laid out for this project but with the following changes to the indicated items:

1. Low-pass filter
2. Butterworth response
3. No passband ripples (Butterworth has none)
4. $f_{-3\text{ dB}} = 1.0$ kHz
5. Op-amps to be type LM741
6. Target stop-band attenuation at 6 $f_{-3\text{ dB}} <$ -60 dB.

Presented below are the highlights of the design procedure to meet his specification. All page references are to the Appendices.

1. **Transfer Function:** Since the response is to be Butterworth and n = 4, then from page B3

$$H_4(s) = \frac{1}{s^2 + 1.8478s + 1} \cdot \frac{1}{s^2 + 0.7654s + 1}$$

   The graphs on page B8 confirm that this transfer function should meet the pass-band and stop-band specs.

2. For each second-order section,

$$T(s) = \frac{H_o \omega_o^2}{s^2 + \frac{\omega_o}{Q}s + \omega_o^2}$$

and since Q does not scale with frequency, and since $\omega_{oA} = 1$ and $\omega_{oB} = 1$,

$$\therefore \mathrm{Q_A} = 0.5412 \text{ and } \mathrm{Q_B} = 1.3065$$

Making the first section the low-Q section:

$$\mathrm{Q_1} = 0.5412 \text{ and } \mathrm{Q_2} = 1.3065$$

3. Similarly, $\mathrm{H_{o1}} = 1$ and $\mathrm{H_{o2}} = 1$

4. Scaling the resonant frequencies, from page B18, $s \rightarrow \frac{s}{\omega_c}$ where

$$\omega_c = 2\pi \cdot 1000\, Hz = 6283\, \frac{rad}{s}$$

This will give:

$$H_4'(s) = \frac{1}{s^2 + 1.8478s + 1} \cdot \frac{1}{s^2 + 0.7654s + 1}$$

$$H_4'(s) = \frac{1}{(\frac{s}{\omega_c})^2 + 1.8478(\frac{s}{\omega_c}) + 1} \cdot \frac{1}{(\frac{s}{\omega_c})^2 + 0.7654(\frac{s}{\omega_c}) + 1}$$

$$H_4'(s) = \frac{\omega_c^2}{s^2 + 1.8478\omega_c s + \omega_c^2} \cdot \frac{\omega_c^2}{s^2 + 0.7654\omega_c s + \omega_c^2}$$

$$H_4'(s) = \frac{3.9476 \cdot 10^7}{s^2 + 1.161 \cdot 10^4 s + 3.9476 \cdot 10^7} \cdot \frac{3.9476 \cdot 10^7}{s^2 + 4.809 \cdot 10^3 s + 3.9476 \cdot 10^7}$$

5. Plot the new transfer function in MATLAB to make sure it has the shape you expect

The denominator is going to be a 4th order function and in order to plot it, it needs to be expanded. You can do this by hand, but it is tedious. If you like, MATLAB will do it for you with the following code:

syms s

expand($(s \wedge 2 + 1.161e4 * s + 3.9476e7) * (s \wedge 2 + 4.809e3 * s + 3.9479e7)$)

ans =

$s \wedge 4 + 16419 * s \wedge 3 + 134787490 * s \wedge 2 + 648191274000 * s + 1558473004000000$

Once you have done this you can plot the result in MATLAB. Here is code to plot this TF:

```
% Set the numerator and denominator of the TF.
% In this case numerator is a constant.
Num = [(3.9476e7)^2] ;
% Denominator = s^4 + 16419*s^3 + 134787490*s^2 + 648191274000*s + 1558473004000000
Dem = [1 16419 134787490 648191274000 1558473004000000];

% Set the frequency range to plot and the value of s.
f = 100:1:1e4;
s = 2*pi*f*j;

% Determine the magnitude of the frequency response in dB.
FR = 20*log10(abs(polyval(Num,s) ./ polyval(Dem,s)));
% Plot the result.
semilogx(f,FR)

xlabel ('Frequency (Hz)')
ylabel ('Amplitude (dB)')
axis ([0 1e4 -90 10])
```

The result will look like this:

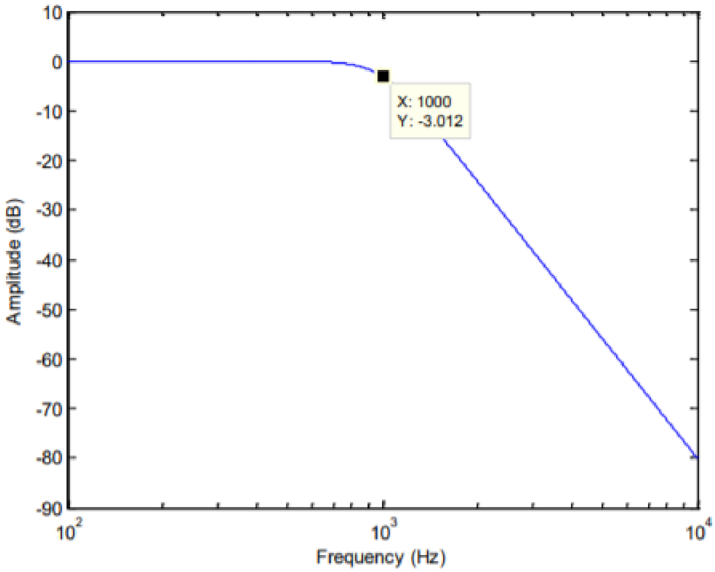

Figure 2: MATLAB Calculated Magnitude Response of a Transfer Function

If we were given the expanded polynomial coefficients and wanted to factor it into second order polynomials, you could use the following code.

```
polyCoeff = [1 16419 134787490 648191274000 1558473004000000];
polyRoots = roots(polyCoeff);

% polyRoots is a vector with the 4 roots of the polynomial. Combine the first
% two and the second two to create two second order polynomials

poly1Coeff = poly(polyRoots(1:2))
poly2Coeff = poly(polyRoots(3:4))
% poly1Coeff=[1 4809 3.9479e7], poly1Coeff=[1 11610 3.9479e7]
```

6. Now that we have calculated to proper transfer function, we can design the circuit. Since one of the transfer functions has a low Q, we will use a Sallen-Key Low-pass circuit which only requires 1 op-amp. The second transfer function has a higher Q portion, so we will use the Tow-Thomas biquad. To reduce clipping, the lower Q circuit will go first:

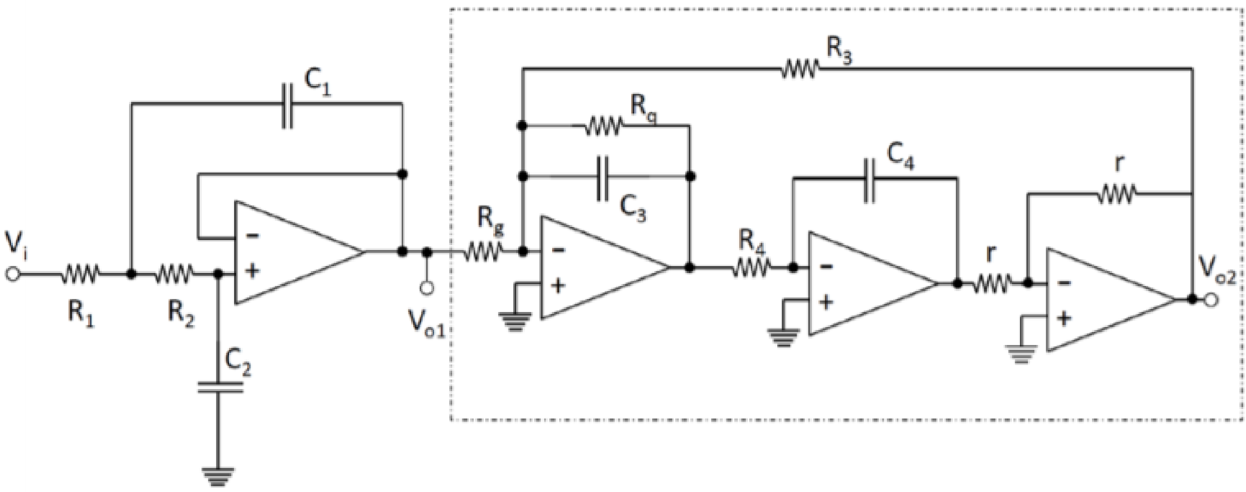

Figure 3: Stallen-Key/Tow-Thomas Biquad Low-Pass Filter

DO NOT PROCEED WITH THE CIRCUIT DESIGN UNTIL YOU KNOW THE TRANSFER FUNCTION IS CORRECT!

Unity-Gain Circuit	Tow - Thomas Biquad
Let $R_1 = R_2$	Let $R_3 = R_4 = R$, $C_3 = C_4 = C$
$\omega_{o1} = \frac{1}{R_1\sqrt{C_1 \cdot C_2}}$	$\therefore \omega_{o2} = \frac{1}{RC}$
$Q_1 = \frac{1}{2}\sqrt{\frac{C_1}{C_2}}$	$Q_2 = \frac{R_q}{R}$
$H_{o1} = 1$	$H_{o2} = \frac{R}{R_g}$

7. **Unity-Gain Circuit Component Values**

Let $R_1 = R_2 = 6.8$ kΩ. (within guidelines)

$$\therefore \sqrt{C_1 \cdot C_2} = \frac{1}{\omega_{o1} R_1} = \frac{1}{6283\,\frac{rad}{s} \cdot 6800\,\Omega} = 23.4\,nF$$

$$Q_1 = \frac{1}{2}\sqrt{C_1 \cdot C_2} = 0.5412$$

$\therefore$ $C_2 = 21.6$ nF and $C_1 = 25.4$ nF ($\gg$ stray capacitances)

Use $C_2 = 22$ nF and $C_1 = 22$ nF $\|$ 3.3 nF

Note: This design started off by assuming resistor values and calculating the desired capacitors. Conveniently, the calculated capacitances are easily obtained with discreet capacitors that are available in the lab. **However, this may not always be the case**. For more flexibility you may want to choose capacitors as it is easier to customize resistors than vice versa.

Luckily $H_{o1} = 1$ (if H_{o1} were <1, R_1 could be replaced by a resistor-divider with attenuation = H_{o1} and a resistance, as seen by the circuit, of R_1.) If we needed to implement such attenuation, we could achieve this by replacing R_1 with two resistors, as seen below.

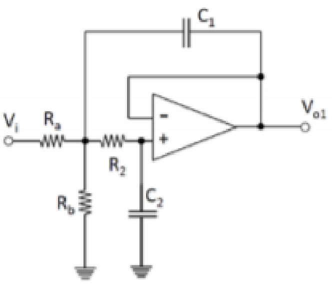

Figure 4: Stallen-Key Low Pass Circuit

The effective value of R_1 is then

$$R_1 = R_a \parallel R_b$$

while the attenuation implemented is equal to

$$H_o = \frac{R_b}{R_a + R_b}$$

8. **Tow-Thomas biquad Component Values**

   Let r = any value within the guidelines. We would like to let C = 22 nF but to relieve pressure on lab supplies, let

   C = 15 nF

   $\therefore \omega_o = \frac{1}{RC} = 6283 \, \frac{rad}{s}$

   $\therefore R = 10.6 \, k\Omega$
   Use R = 10 kΩ + 560 Ω.

   $\therefore Q_2 = \frac{R_q}{R} = 1.3065$

   $\therefore R_q = 13.9 \, k\Omega$
   Use R_q = 10 kΩ + 3.9 kΩ.

   $\therefore H_{o2} = \frac{R}{Rg} = 1$
   Use R_g = 10 kΩ. + 560 Ω
   (Resistor values are all within the guidelines and capacitor values >> stray capacitances.)

9. All op-amps are type LM741CN (from page B22)

   Do not forget to connect all of the op-amps to ±15 V! This is a very common mistake.

10. The design is now complete. **After completion, each second-order section is then assembled and independently tested and adjusted to confirm that H_o, Q and w_o of each part by itself matches the expected values**. Once properly adjusted, the two sections can be connected together to produce the desired transfer function (Butterworth for the above design). The independent adjustment procedure is used because it is nearly impossible to simultaneously tune all the sections of a multiple section filter.

Note, the above is for your information and so you can see the steps in doing a filter design. You can follow similar steps when designing your band-pass Chebyshev filter, but of course the transfer function and pole locations will be very different.

Additional filter design information will be given in a lecture prior to the lab.

DAY 1: CHEBYSHEV FILTER DESIGN AND SIMULATION, START OF CONSTRUCTION

***** Very Important *** Note, the paper design is to be done prior to the lab *****

The design of the Chebyshev filter will be somewhat similar to the design done above. A teaching assistant will check out your design calculations, and you will run a simulation of the filter. The purpose of running a simulation is to catch design errors early before you spend a lot of time building it.

The first step in the design will be coming up with a transfer function to implement. This won't be trivial, but there are many useful hints in the lab appendix. Remember that you need to break your filter up into second order transfer functions in order to implement it.

Once you have done that you need to make a circuit implementation of your transfer function. After selecting your resistors and capacitors, simulate your filter's frequency response. After a successful simulation, **export your simulation data** so that it can be plotted directly against your measured data in your report.

Do NOT wait until Day 2 to begin filter construction. The circuit will be very complicated and will need debugging. Have your circuit ready for the beginning of Day 2.

DAY 2: CHEBYSHEV FILTER DEMONSTRATION

During this lab period, students will present their final working filters and demonstrate that they meet all of the stated requirements. Only after a successful demonstration will a teaching assistant sign a student's schematic. Students are reminded to have their circuits working as soon as possible in the lab period as there may be a rush of students at the end all looking for a demo.

Measurements

Once you have built your circuit and confirmed that your measured values of ω_o, Q, a_1 etc. are similar to what you expected them to be, you should confirm that the entire circuit meets all of the specifications listed.

1. Find $f_{3\text{ dB}}$ by locating the frequency where the gain drops to -3 dB for the last time at each edge of the passband.
2. Confirm the pass-band ripple meets specifications by looking at the gain for every frequency within the passband. The gain should be within 0 to -3 dB with 1 dB of margin (i.e. between -4 to 1 dB).
3. **For EACH 2$^{\text{nd}}$ order section and the overall filter**, measure and plot gain as a function of frequency and compare to your original polynomial AND simulated results. You should have at least 30 points from $0.1^*f_{-3\text{ dB lower}}$ to $10^*f_{-3\text{ db upper}}$.
4. Take more measurements around the regions of interest (i.e. the passband ripples and -3 dB points).
5. For interest you may hook your circuit input up to a signal source from a computer and the output to a speaker, similar to lab 2. Do this to verify that only the range of frequencies designed are heard out of the speaker.

Report Requirements

Your report will need to clearly document the steps you went through to design your filter. The report itself should clearly show the major steps you went through, although you may put the complete calculations in the appendix. Even if you do so, you still need to clearly discuss what you did in the reports (for instance, your report can have the starting point and the final results for a major step in the calculations, with a description of what went on in between the two, and with the full calculations in the appendix).

The report will also need a discussion about the simulation of the filter and the steps you underwent to implement your filter. If you made any changes to adjust the gain/Q/frequency, you should show what you did and why.

The report will compare the filter specs, the theoretical response, and the simulated results. Thus, you will need a plot showing the amplitude vs. frequency for both filter sub-modules and for the entire filter (3 plots in total). Each plot must show the theoretical response (from your transfer function), and the simulated response **on the same chart**. The plot for the overall filter response should also show lines indicating the minimum and maximum allowable pass-band gain and minimum and maximum allowable $f_{-3\text{ db}}$ corners, as per the specifications. The theoretical, simulated and measured responses should all fit nicely within these boundaries. An example is found in the sample chart for lab 2. The point of these charts is to show very clearly that your simulations match your theoretical values, and that both satisfy the specifications for your circuit. Note that to properly show high and low amplitudes, you will need to plot gain in dB, and to properly show low and high frequency response, frequency should be in a logarithmic scale.

For each sub-module and for the entire circuit, you will need at least 30 points to get a smooth curve that properly shows the shape of the peaks. This process can take an hour or longer, so do not leave this to the last moment. As you might expect, you should not take these measurements until you have verified that your entire filter meets all specifications.

The gain and phase plots for an example 4th order Chebyshev low-pass filter is provided in Appendix A. Additional examples and descriptions for filter design is given in Appendix B.

Other Notes on Meeting Filter Specifications

1. As capacitor values and dissipation factors change slightly with the magnitude of the applied waveform, try to tune and demonstrate your filter at the same input amplitude setting.
2. Be sure to understand the difference between the transfer function and the circuit:
   (a) The transfer function often follows some well-known characteristic response such as Bessel, Butterworth, Chebyshev, Cauer, or Thompson.
   (b) To obtain the desired transfer function, second-order filter sections are often cascaded. The sections are the circuits, many of which are well-known, such as Sallen-Key, Thomas Biquad, unity-gain, and 100's more.

3. When measuring a second order transfer function, there are some short-cuts that can be done to quickly determine the key parameters (H_o, Q, ω_o, etc.) of it. The exact steps depend on what type of transfer function you are implementing. Typically, this will involve plotting a Lissajous figure (explained later) and sweeping the frequency to find when the phase shift is ± 0, 90 or 180 degrees, and then measuring the gain at this frequency, at multiples of this frequency, and at other identifiable points (such as when the gain is the highest/lowest). The parameters are then found from calculations from the measured gain and frequency values.

   Consider this chart showing the parameters for a low pass circuit:

$$T(S) = \frac{H_o \omega_o^2}{S^2 + S\frac{\omega_o}{Q} + \omega_o^2}$$

(Phase shift at ω_o = - 90°)

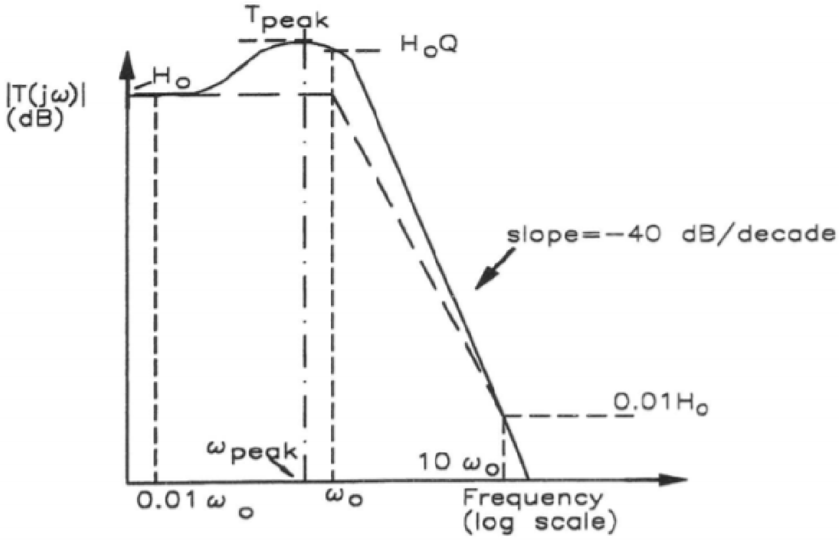

Figure 5: Low Pass Circuit Parameters

   By finding ω_o (which is done by looking for a -90 degree phase shift using the Lissajous figure), we can then find H_o and Q by making appropriate measurements of the gain at appropriate frequencies. By measuring the gain at 10*ω_o, we can also verify that the circuit is implementing a second order low-pass function.

   A high pass function is the same, but reversed in the frequency axis.

   Just about everything you could ever want to know about a second-order BPF is shown in Figure 6. **Note that at ω_o the phase shift will be either 0 or 180 degrees depending on the sign of a_1. This is different than a LPF where the phase shift is 90 degrees.**

   To measure a circuit implementing a 2$^{\text{nd}}$ order band-pass transfer function, we would set up the oscilloscope to plot a Lissajous figure and adjust the frequency to get a straight line (which indicates a 0 or 180 degree phase shift, as opposed to a circle or flat/tall ellipse, which indicates ± 90 degrees). That frequency reading is ω_o. Then we can determine the 3 dB bandwidth of your circuit. Then Q is given by:

$$Q = \frac{\omega_o}{BW} \tag{0.0.1}$$

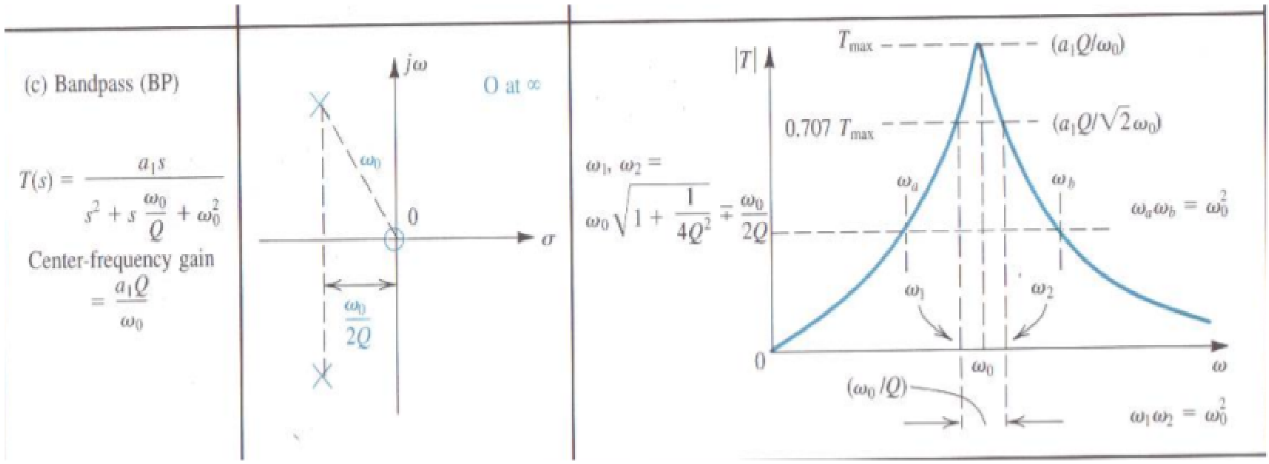

Figure 6: Band Pass Second-Order Filter Parameters

Then measure T_{max}. Knowing Q you can determine a_1. Next compare your values of ω_o, Q, and a_1 to your theoretical and simulated values. If they are similar, keep going. If they are significantly different, you will need to figure out why.

When first constructing your circuit, follow this procedure to get very quick readings of ω_o, a_1, Q, etc. for both 2^{nd} order transfer function circuits individually (disconnect them from each other). Adjust the peak-to-peak value of the input voltage to a convenient value. This is needed to quickly verify that each circuit is within parameters. For your report, you will need to take detailed measurements of both but do not do these until you verify your entire circuit is working.

Lissajous Figures

Under normal circumstances, the screen on the oscilloscope plots the data received from the probes as a function of time. However, for a Lissajous figure, we will plot 1 channel as a function of the other. Thus the input of your filter will be plotted along the y-axis and the output along the x-axis (or the other way around).

This makes it easy to determine if the phase between the input and output is certain key values. For instance, at 0° or 180° the Lissajous figure will be a straight line at an angle determined by the gain. Alternatively, when the phase shift is exactly 0° or 180°, the display will show a symmetrical ellipse. Here, the major axes of the ellipse will coincide with the axes of the display. A very small change of frequency causes the result to be a slanted ellipse, so it is quite easy to get very close to the frequency ω_o.

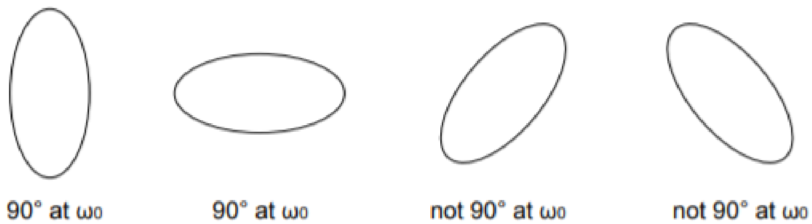

Figure 7: Lissajous Pattern Figures

To do this, do the following:

1. Ensure that both oscilloscope inputs from the Multisim oscilloscope are connected to the desired locations on the sub-module.
2. Ensure that at least three complete cycles are visible on the display and the magnitude scale should cause the waveforms to fill as much of the screen without clipping.
3. In the bottom right corner of the oscilloscope window is the Timebase settings. Here, the default display setting is Y/T, or amplitude with respect to time. For a Lissajous figure, change the display setting to A/B.

4. Once you have determined where ω_o is, you no longer need the A/B mode, and you should move back to the Y/T mode for your measurements.

Note that using the Lissajous figure only makes sense for the 2^{nd} order sub-modules individually. For the entire filter circuit, you must use the YT mode and find $f_{3\ dB}$ the normal way by looking for where the gain drops to -3 dB for the last time.

Sensitivity Analysis

Page B1 and B2 tells us how the desired parameters of the filter (ω_o, Q , H_o) will change if we change the component values. This is very useful for correcting errors. For example, if the pole frequency is measured to be off by 5%, sensitivity analysis can tell us by what percentage we should change a particular component value to bring the pole frequency back to the desired value. Thus, Sensitivity is measured as $\frac{\Delta\omega_o}{\Delta R}$ or $\frac{\Delta Q}{\Delta C}$. There is obviously some math involved, but the result is very straight forward and it ends up that the sensitivity is equal to the exponent in the equation relating the parameter to the component. Here are a few examples:

$\omega_o = \frac{1}{RC} = R^{-1}C^{-1}$ so $\frac{\Delta\omega_o}{\Delta R} = -1$. This means to change ω_o by +3% requires a change of R by –3%.

$Q = \sqrt{\frac{C_1}{C_2}} = C_1^{\frac{1}{2}}C_2^{-\frac{1}{2}}$ so $\frac{\Delta Q}{\Delta C_1} = \frac{1}{2}$ and $\frac{\Delta Q}{\Delta C_2} = -\frac{1}{2}$. Thus if Q is high by 5%, we would like to decrease it by 5%, thus we want ΔQ of –5% which we can achieve by decreasing C_1 by 10% or we by increasing C_2 by 10%. We could see this directly from the equation, or from the sensitivity equation.