EXPERIMENT NUMBER 8
Introduction to Active Filters

Preface:
- Preliminary exercises are to be done and submitted individually.
- Laboratory hardware exercises are to be done in groups.
- This laboratory requires technical memorandum to be submitted individually. The technical memorandums require a specific format, must include specific appendix tables, and must address the listed questions. Review the associated guidelines.
- The laboratory notebooks should include all settings, steps and observations in the exercises. All statements must be in complete sentences and all tables and figures must have a caption.
- Review the guidelines for plagiarism to be aware of acceptable laboratory and classroom practices.

Filters are fundamental to many circuit designs and they exist for analog and digital applications. Applications include noise reduction in communication systems, band-limiting of signals before sampling them, conversion of sampled signals into continuous-time signals, signal demodulation, improving the sound quality of audio system components such as loudspeakers and receivers, and many others.

Objectives:
- To learn how to create filter circuit models in PSpice.
- To learn how to simulate circuit models using AC Sweep Analysis.
- To learn how to construct active low-pass, high-pass and band-pass filter circuits.
- To determine the frequency responses of active low-pass, high-pass and band-pass filters.

References:
- PSpice experiments in EE 152 and PSpice Tutorials (See departmental website)
- EE 151 and EE 153 text: Cunningham and Stuller, *Circuit Analysis*, 2nd Ed. (Houghton Mifflin Company, Boston, 1995).
- Neamen, Donald A., Electronic Circuit Analysis and Design, 2nd ed., (McGraw-Hill, New York, New York, 2001), Chap 15.

Background:
Filters are used in circuits to block undesired frequencies and there are two main types: (1) active and (2) passive. The most common filters are low-pass, high-pass, band-pass and band-reject. Each filter has a specific cut-off frequency that is determined by resistor and capacitor values in the circuit. The cut-off frequency is determined by equation 1. The cut-off frequency is often referred to as the 3dB cut-off and is when the output has an amplitude of $1/\sqrt{2}$ or 0.707 times the maximum input.

$$F_c = \frac{1}{2\pi RC} \qquad (1)$$

Consider the low-pass filter, all frequencies below the cut-off are passed at maximum value and slowly begin to decline as the cut-off frequency is approached. At the cut-off frequency, the output ideally has an amplitude of $1/\sqrt{2}$ or 0.707 times the maximum input. After the cut-off frequency the output continues to decline with the same slope as before until reaching zero. The opposite can be said for the high-pass filter; low frequencies are blocked until the cut-off frequency is reached. The band-pass filter is does exactly what its name implies, frequencies within a specified bandwidth are passed and all others are rejected. The band-reject filter works in an opposite manor from the band-pass filter and is sometimes referred to as a notch filter.

Passive filters, as shown in Figure 1, contain only passive elements such as, resistors, capacitors and inductors and generally provide a maximum gain of 1. Furthermore, when an impedance is added in series or in parallel to the load, the output amplitude is directly affected and the filter must be redesigned. This is referred to as a loading affect, which may change the low and/or high cutoff frequencies. The only passive filter that can amplify its output is the RLC resonant filter. To avoid redesigning the filter for each application, active filters are used. An active filter simply implies that an active device is used in the circuit, such as an op-amp as shown in Figure 2. Active filters allow for a gain greater than 1 and the loading effect is minimal, meaning that the output response is essentially independent of the load driven by the filter.
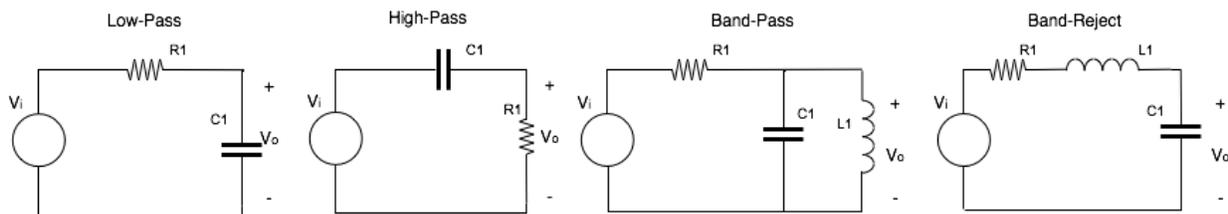
**Figure 1: Passive filter circuits**
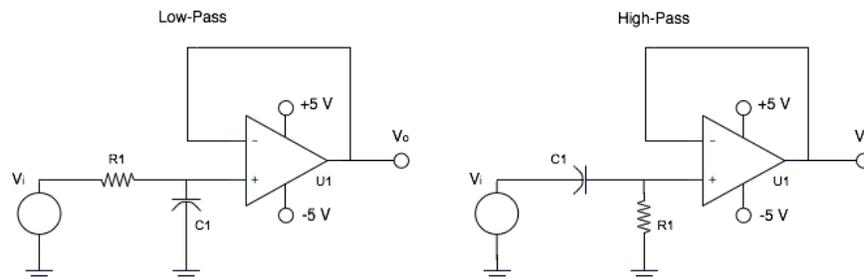
Figure 2: Active filter circuits
(**could add more active filter circuits**)

Band-pass and band-reject filters have two cut-off frequencies, which can be used to calculate the filters' bandwidth. The bandwidth (BWD) is just the high cut-off frequency (f_{HC}) minus the low cut-off frequency (f_{LC}), as shown in equation 2. The high and low cut-off frequencies are calculated from RC pairs within the circuit. Active band filters also have a predictable point where the output will be at its maximum within the bandwidth. The maximum output occurs at the center frequency (f_o). The center frequency calculation is shown in equation 3.

$$BWD = f_{HC} - f_{LC} \tag{2}$$

$$f_o = \sqrt{f_{LC}f_{HC}}$$ (3)

As a reminder the pin out diagram of the op-amp is provided in Figure 3. The input terminals of the OpAmp are pin 3 - (+) or non-inverting, and pin 2 - (-) or inverting. A positive voltage up to15V should be applied to the (+) terminal and negative voltage down to -15V should be applied to the (-) terminal. Pin 6 is the output of the op-amp. The unmarked pins are not used in this experiment. Pins 1 and 5 are for the offset null and pin 8 is not connected.
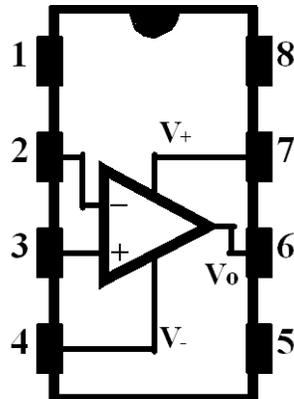
Figure 3: Op-amp pin out diagram

Preliminary: Revised: Spring 2013

(NOTE: The sections in red highlight the three printouts that are to be turned in as the prelim for lab 8.)

1. Start MATLAB

2. Open a new Simulink window by clicking on the Simulink button. The simulink button is in the top left of the MATLAB window. The Simulink icon is highlighted in Figure 1 shown below.
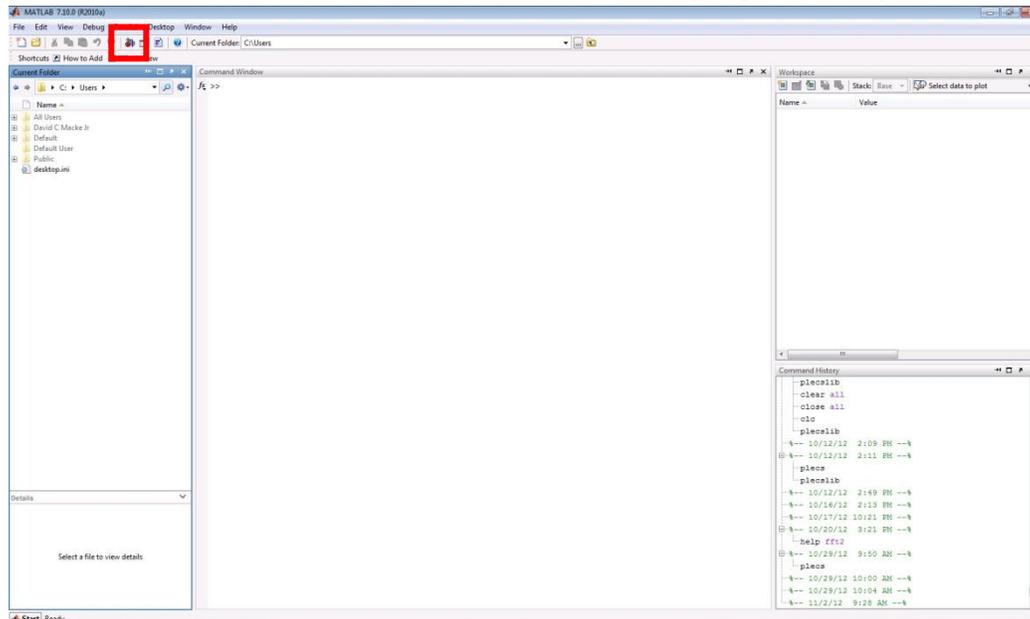

Figure 1:MATLAB Start Page

3. The window that appears is called the Simulink Library Browser. This window contains every component that can be placed into a simulation. The Library Browser window can be searched in two ways, with the search box at the top of the window or with the explorer window to the left of the window. The search bar, highlighted in Red, and the Explorer Window, highlighted in Green are shown in the image below. Create a New Model, click on the New Model Button. The New Model button is located in the top left of the Simulink Library Browser window. The New Model button, highlighted in Blue in Figure 2, is shown below.
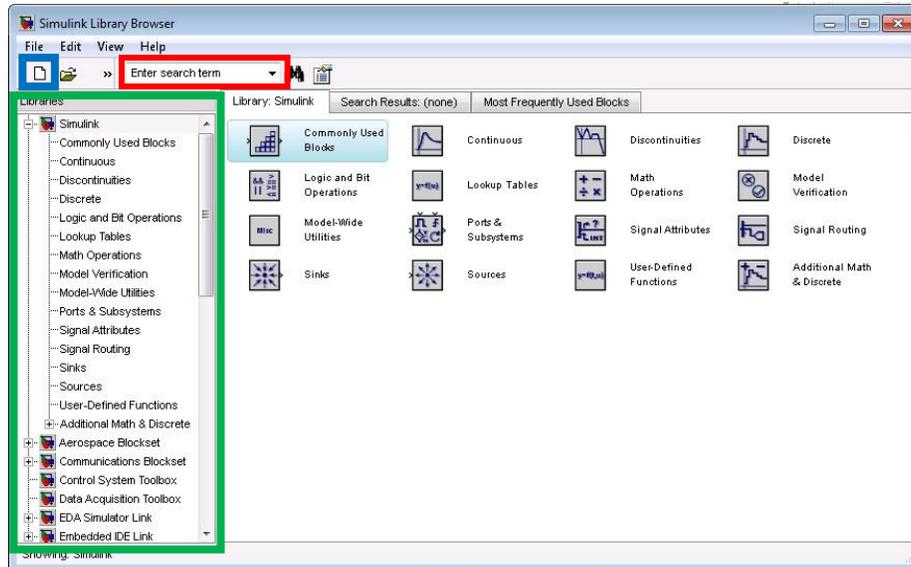
Figure 2: Simulink Library Browser

4. After the New Model window opens up, click on the Simulink Library Browser window. In the Library Browser, look at the Libraries window on the left. Scroll down until the PLECS Library appears. Click on the PLECS library, Drag and Drop the Circuit component into the New Model Window. In the Model window, double click on the PLECS Circuit name "Circuit" and change the name to "Part 1".
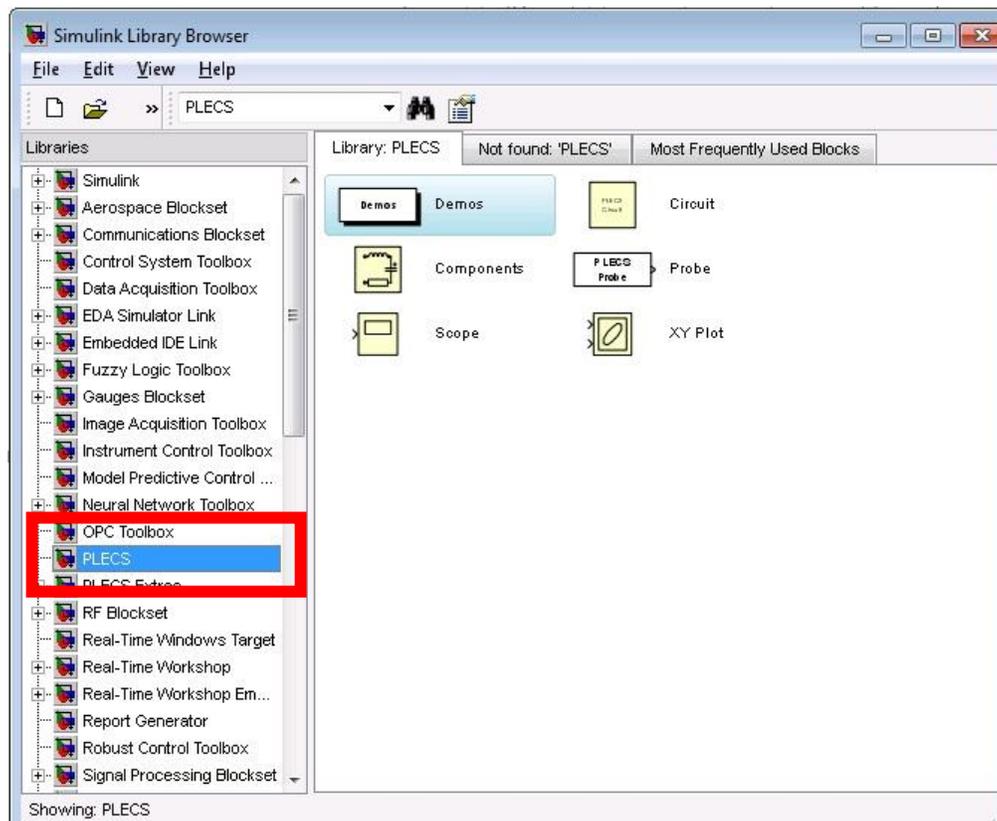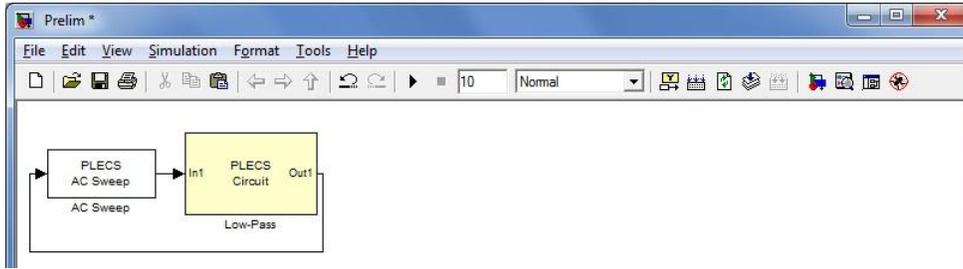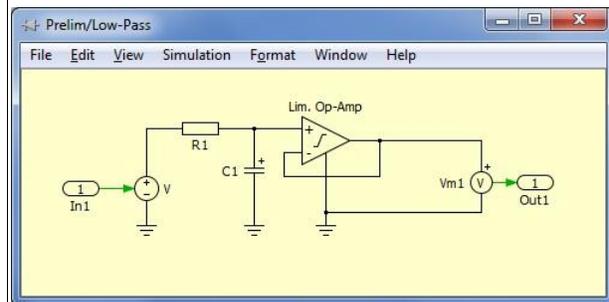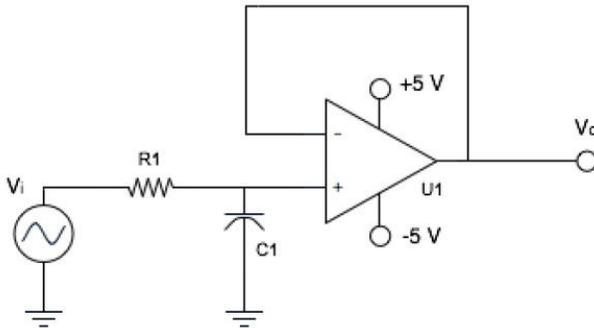

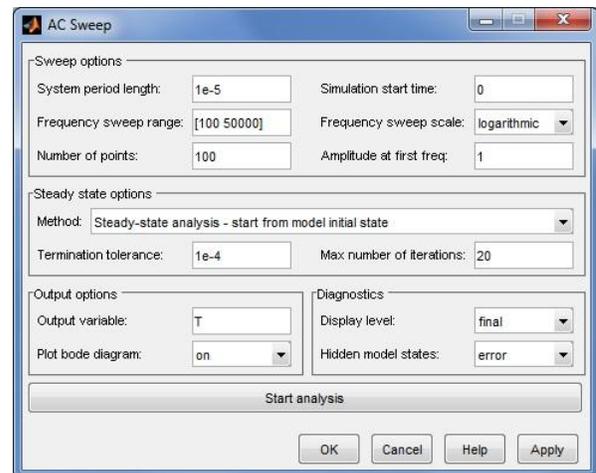**Figure 3: Simulink Library Browser - PLECS Library**

Part #1

5. Use PLECS to simulate the filter shown in Figure 4. Perform an AC sweep to observe the frequency response of the filter and the ideal cut-off frequency. First Navigate to the PLECS Extras and place an AC Sweep block. Next drag a circuit block from the PLECS Library. Double click the Circuit block. Within the PLECS Circuit build the filter shown below. First place a Voltage Source (Controlled), a resistor, a capacitor, three grounds, and a voltmeter. The Resistor should be set to 33kΩ and the capacitor to 4.7nF. Next place an Op-Amp with Limited Output. The Component can be found in the PLECS Library under Electrical->Electronics. After placing the Op-Amp change the Open-loop gain to 2e6 and the Output voltage limits to [-5 5]. Finally place a Signal Inport and connect it to the Voltage Source and place a signal Outport and connect it to the Voltmeter. The result should look like the circuit below. In the Simulink window connect the right side of the AC Sweep block to the input of the PLECS Circuit and connect the output of the PLECS circuit to the left of the AC Sweep block.

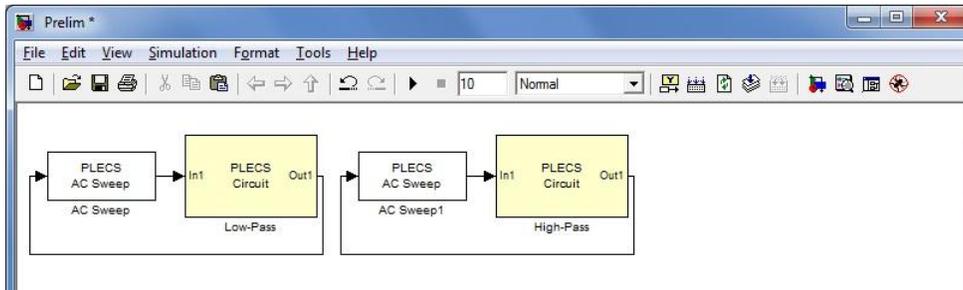6. Now setup the AC Sweep block. Double click on the AC Sweep to bring up the block parameters. Change the block to have the following parameters.

System Period Length:1e-5   Simulation Start Time:0
Frequency Scale: Logarithmic
Frequency Sweep Range:100Hz to 50kHz [100 50000]
Number of Points:100
Amplitude at First Freq: 1
Method:Steady-state Analysis – Start from model
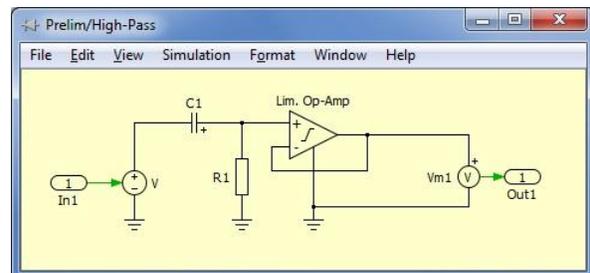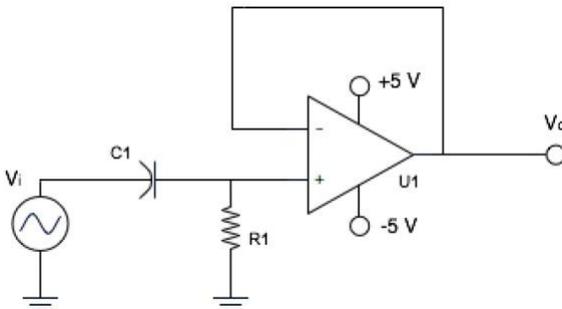Plot bode diagram: On

7. Push the Start Analysis Button

8. Make a print out of the frequency response, calculate the cut-off frequency of Filter 1 using equation 1 and mark it on the print.
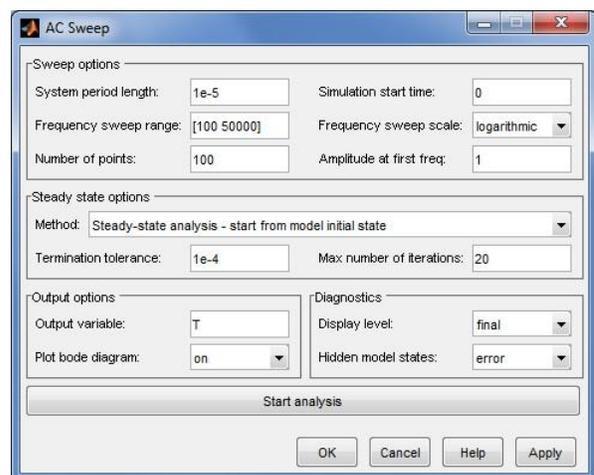.

Part #2

9. Use PLECS to simulate the filter shown in the figure below. Perform an AC sweep to observe the frequency response of the filter and the ideal cut-off frequency. First Navigate to the PLECS Extras and place an AC Sweep block. Next drag a circuit block from the PLECS Library. Double click the Circuit block. Within the PLECS Circuit build the filter shown below. First place a Voltage Source (Controlled), a resistor, a capacitor, three grounds, and a voltmeter. The Resistor should be set to 33kΩ and the capacitor to 4.7nF. Next place an Op-Amp with Limited Output. The Component can be found in the PLECS Library under Electrical->Electronics. After placing the Op-Amp change the Open-loop gain to 2e6 and the Output voltage limits to [-5 5]. Finally place a Signal Inport and connect it to the Voltage Source and place a signal Outport and connect it to the Voltmeter. The result should look like the circuit below. In the Simulink window connect the right side of the AC Sweep block to the input of the PLECS Circuit and connect the output of the PLECS circuit to the left of the AC Sweep block.

10. Now setup the AC Sweep block. Double click on the AC Sweep to bring up the block parameters. Change the block to have the following parameters.

System Period Length:1e-5   Simulation Start Time:0
Frequency Scale: Logarithmic
Frequency Sweep Range:100Hz to 50kHz [100 50000]
Number of Points:100
Amplitude at First Freq: 1
Method:Steady-state Analysis – Start from model
Plot bode diagram: On

11. Push the Start Analysis Button

12. Make a print out of the frequency response, calculate the cut-off frequency of Filter 2 using equation 1 and mark it on the print
.

Part #3
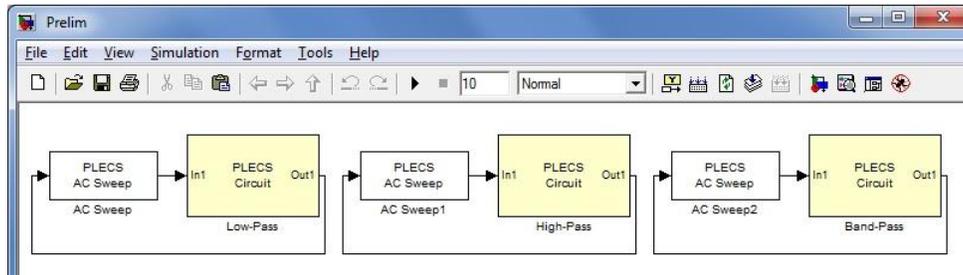
13. Use PLECS to simulate the filter shown in the figure below. Perform an AC sweep to observe the frequency response of the filter and the ideal cut-off frequency. First Navigate to the PLECS Extras and place an AC Sweep block. Next create the filter shown below with the following parameters. Set R1=22k, R2=470, C1=4.7n and C2=0.01u. Place a voltage probe on the source and the output. Next, open the analysis set-up window and choose AC Sweep. Make a print out of the frequency response and calculate the cut-off frequency of each individual filters within Filter 3 using equation 1, the filter bandwidth using equation 2 and the center frequency using equation 3. Mark the center frequency on the print out of the waveform.
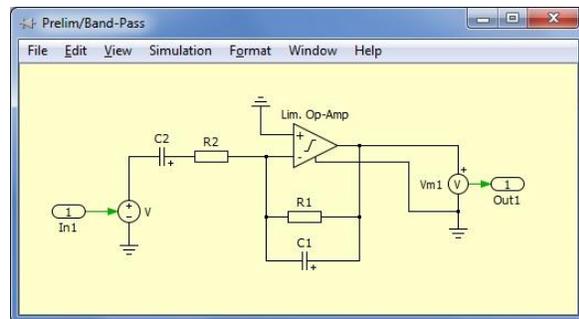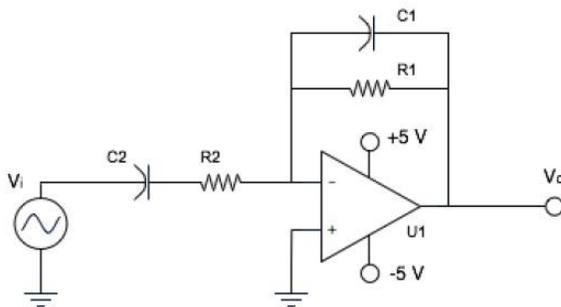
Note: The AC Sweep block should be set like the previous steps.

Experimental Procedure:
(Record specifics in the Laboratory Notebook.)

1. Build the circuit in Figure 4 from the preliminary on a breadboard. Use the same component values of R=22kΩ and C=.0047uF. Use a power supply to power the op-amp with +5 and -5 volts and connect a function generator as the circuit input. ***Do not use the same ground connect for Vi and the op-amp, your readings will be incorrect. Set the function generator to 2 V_{pk-pk} sine wave starting at a frequency of 1Hz. Use the oscilloscope to observe the output of the circuit and the voltage output of the function generator (use a "T" connector at the function generator output). Record the initial peak-to-peak output of the circuit (V_{pk-pk}).

    Q1: What is the initial output value of the circuit?

    Vary the frequency of the function generator and fill in the values for Filter 1 in Table 1.

Table 1: Experimental frequency response values

F (Hz)	Filter 1 V_{pk-pk}	Filter 2 V_{pk-pk}
100		
300		
500		
700		
900		
1k		
1.1k		
1.2k		
1.3k		
1.4k		
1.5k		
2k		
5k		
10k		
15k		
20k		
30k		
40k		
50k		

> *Q2: What is the experimental cut-off frequency and how does it compare to the theoretical value?*
>
> *Q3: Calculate the percent difference between the theoretical and experimental cut-off frequencies?*

2. Build the circuit in Figure 6 from the preliminary on a breadboard. Use the same component values of R=22kΩ and C=.0047uF. Use a power supply to power the op-amp with +5 and -5 volts and connect a function generator as the circuit input. ***Do not use the same ground connect for Vi and the op-amp, your readings will be incorrect. Set the function generator to 2 V_{pk-pk} sine wave starting at a frequency of 1Hz. Use the oscilloscope to observe the output of the circuit and the voltage output of the function generator (use a "T" connector at the function generator output). Record the initial peak-to-peak output of the circuit (V_{pk-pk}).

    Q4: What is the initial output value of the circuit?

Vary the frequency of the function generator and fill in the values for Filter 1 in Table 1.

Q5: What is the experimental cut-off frequency and how does it compare to the theoretical value?

Q6: Calculate the percent difference between the theoretical and experimental cut-off frequencies?

3. Reconstruct Filter 1 and change the input to a 2 V_{pk-pk} square wave at 1Hz. Use the oscilloscope to observe the input (function generator) and output (pin 6 of op-amp) waveforms on the same graph. Align the waveforms so they overlap. Vary the frequency of the function generator between 1 and 500Hz. Do you notice a relationship during low-high and high-low transitions of the square wave? Record the graph on the oscilloscope with the function generator set to 100Hz.

Q7: What relationship do you notice in the frequency response?

4. Reconstruct Filter 2 and change the input to a 2 V_{pk-pk} square wave at 1Hz. Use the oscilloscope to observe the input (function generator) and output (pin 6 of op-amp) waveforms on the same graph. Align the waveforms so they overlap. Vary the frequency of the function generator between 30k and 50kHz. Do you notice a relationship during low-high and high-low transitions of the square wave? Record the graph on the oscilloscope with the function generator set to 50kHz.

Q8: What relationship do you notice in the frequency response?

5. Build the circuit in Figure 7 from the preliminary on a breadboard. Use the same component values of R1=22kΩ, R2=470Ω, C1=4.7nF and C2=0.01uF. Use a power supply to power the op-amp with +5 and -5 volts and connect a function generator as the circuit input. ***Do not use the same ground connect for Vi and the op-amp, your readings will be incorrect. Set the function generator to 2 V_{pk-pk} sine wave starting at a frequency of 1Hz. Use the oscilloscope to observe the output of the circuit and the voltage output of the function generator (use a "T" connector at the function generator output). Record the initial peak-to-peak output of the circuit (V_{pk-pk}). Vary the frequency of the function generator and record the V_{pk-pk} output for enough points to reconstruct a graph reflecting the cut-off frequencies and the overall frequency response. Can you find the maximum output?

Q9: What is the experimental center frequency and how does it compare to the theoretical value? What are F_{CL} and f_{HC}?

Q10: Calculate the percent difference between the theoretical and experimental center frequencies?

Technical Memorandum:

- Memorandum discussion:

1) Discuss your observations for Filter 1. What type of filter is Filter 1? Was the initial output value as you expected? Explain. (Q1) Construct a graph with the values collected in Table 1 for Filter 1 and compare that graph with the simulation results from the preliminary. Was the cut-off frequency similar? (Q2) Was the slope of the curve similar? Explain any discrepancies. (Q3)

2) Discuss your observations for Filter 2. What type of filter is Filter 2? Was the initial output value as you expected? Explain. (Q4) Construct a graph with the values collected in Table 1 for Filter 1 and compare that graph with the simulation results from the preliminary. Was the cut-off frequency similar? (Q5) Was the slope of the curve similar? Explain any discrepancies. (Q6)

3) Discuss your observations of filters with a switching input. What do you notice? (Q7 and Q8)

4) Discuss your observations for Filter 3. What type of filter is Filter 3? Was the initial output value as you expected? Explain. Construct a graph with the collected V_{pk-pk} values for Filter 3 and compare that graph with the simulation results from the preliminary. Was the center frequency similar? (Q9) Were the cut-off frequencies similar to the theoretical values? (Q9) Explain any discrepancies. (Q10)

- Appendix 1: Plot the output voltage curve vs. frequency for Filter 1. Specify the cut-off frequency on the plot (Q2).

- Appendix 2: Plot the output voltage curve vs. frequency for Filter 2. Specify the cut-off frequency on the plot (Q5).

- Appendix 3: Plot or sketch the resultant oscilloscope waveforms for a square wave input to Filters 1 and 2.

- Appendix 4: Plot the output voltage curve vs. frequency for Filter 3. Specify the lower and upper cut-off frequencies on the plot and the center frequency (Q9).