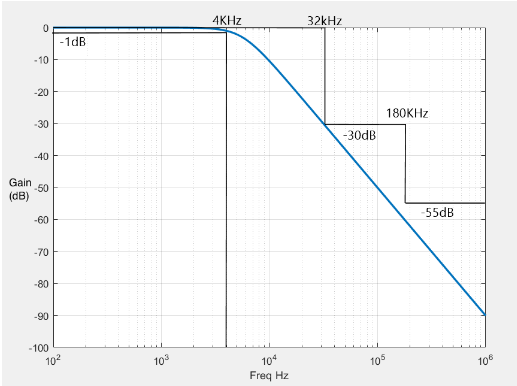
UC San Diego, ECE 100

Lab Project 2: Active Circuit Design

This lab project aims to design and simulate the low-pass filter needed to separate the digital DSL signal from the voice signal on a telephone line. The specifications for this filter are sketched below.



1. **Approximation:** The basic objective is (always) to realize the circuit as economically as possible. The cost of a circuit depends on the "order" because the higher the order, the more components you need. So the first step in any filter design is to find the lowest order rational function of s that will meet the spec. In this case, it is easy to see by eye that we will need a transfer function that falls at least as fast as 40 dB/decade, so we start by checking to see if a second-order filter will work. We start with a Butterworth filter for convenience.

   (a) The magnitude squared of the frequency response of a Butterworth filter is $|H(f)|^2 = \frac{1}{1+(\frac{f}{f_0})^4}$. To see if the transfer function meets the spec, we choose f_0 so the transfer function just hits the passband edge. This is the lowest possible value of f_0. Then we check if $|H(f)|^2$ clears the stopband edge. Find this value of f_0 analytically. Calculate the resulting $|H(f)|^2$ at 32 and 180 kHz. Does the filter meet the spec?

   (b) You will find that the maximally flat transfer function (the Butterworth filter's characteristic) meets the spec, but it is a very tight fit. This leaves little room for component errors. We could increase the order from $n = 2$ to $n = 3$, but this will increase the cost by roughly 50%. So instead, we will design and use another filter transfer function. We will allow the $|H(f)|^2$ in the passband to "ripple" from -1 dB at dc up to 0 dB, then down to -1 dB at 4 kHz. Thus, $|H(f)|^2$ will have a peak 1 dB greater than the dc value. At dc, $10 \log |H(f = 0)|^2 = -1dB$, which means $|H(f = 0)|^2 = 0.7943$.

Following the standard transfer function for a second-order low-pass filter with DC gain of $-1dB$, $H(s) = \frac{(0.7943)^{\frac{1}{2}}}{1+\frac{2\zeta s}{\omega_0}+\frac{s^2}{\omega_0^2}}$, find an expression for the frequency at which $|H(j\omega)|$ peaks. Also, find an expression for $\frac{|H(f_{peak})|^2}{|H(0)|^2}$ as a function of ζ. Use MATLAB to plot $\frac{|H(f_{peak})|^2}{|H(0)|^2}$ in dB vs ζ in the range $0.1 < \zeta < 0.7$ and find the value of ζ that will provide a 1 dB peak.

(c) Now that we have the necessary ζ, we can correct the dc gain using a voltage divider to get $|H(f)|^2 = \frac{0.7943}{(1-(\frac{f}{f_0})^2)^2+(2\zeta\frac{f}{f_0})^2}$. All that remains is to choose the natural frequency f_0 so $|H(f)|^2 = -1$ dB at 4 kHz. It is not easy to invert this equation for f_0, although MATLAB can do that. One possible way to find f_0 is as follows. Guess f_0, say $f_0 = 5000$ Hz, and plot $|H(f)|^2$ in dB vs. f over the range $1,000 < f < 10,000$. Find the frequency at which $|H(f)|^2$ drops to -1 dB, say this value is x. We therefore need to scale f_0 by the factor $\frac{4000}{X}$, i.e. $f_0 = 5000 \times \frac{4000}{X}$. Find the required f_0 and plot $|H(f)|^2$ in dB vs. f for $200 < f < 1$ MHz on a semilogx plot. The resulting function will drop faster in the transition band and meet the spec with more clearance than the Butterworth approximation. There is no need to save a copy of this plot.

(d) Finally, we need to adjust f_0 upwards a bit so $|H(f)|^2$ clears the passband edge and the stopband edge by the same factor. What is your final value of f_0? Save a copy of this graph to show that it meets the spec.

Hint: you can check the distance between the passband's edge and the stopband's edge in both frequency and gain.

Option 1: check the distance in gain	Option 2: check the distance in frequency
(-1 dB) - Gain @ 4 kHz = X1	(f @gain = -1 dB)/4 kHz = X1
Gain @ 32 kHz - (-30 dB) = X2	32 kHz/(f @gain = -30 dB) = X2
$\|X1–X2\| < 0.1dB$ is acceptable	$\frac{\|X1-X2\|}{X1} < 10\%$ is acceptable

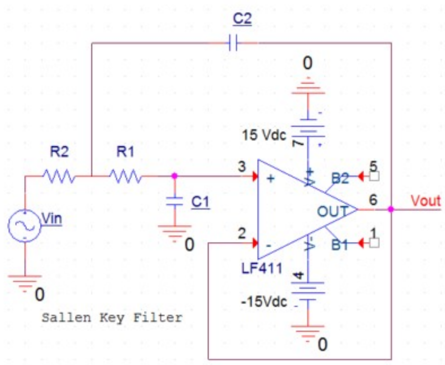
2. **Realization/Simulation:** This filter can be realized with a passive RLC or active RC circuit. The most attractive topology for an active RC low-pass filter is the Sallen-Key form shown on the next page. However, the Sallen-Key circuit has a dc gain of unity, so we'll need to modify it slightly. The circuit components are not ideal, and we must simulate the realization to ensure that non-ideal effects don't interfere with the design.

(a) Analyze the Sallen-Key circuit as shown on the following page. Put the transfer function in the general form. For simplicity, we usually set $R1 = R2$. Assume that the resistors are

$R1 = R2 = 100\ \text{k}\Omega.Calculatethecomponentvaluesneededtoobtainthenecessaryvaluesof\mathrm{f}_0$ and ζ. You found the necessary f_0 and ζ in parts 1(d) and 1(b), respectively. Finally, we can correct the dc gain, setting it to -1 dB, by converting R2 into a voltage divider with a dc gain of -1 dB and a Thevenin resistance of 100K. Do this too.

(b) The most important "non-ideal" opamp effect for this circuit is the output resistance of the opamp itself. This is typically about 50Ω. At the highest frequencies, you can assume that the capacitors are short circuits and the opamp gain is zero. Show that under these conditions $|H| \to \frac{R_{OUT}}{R}$. (Assume $R1 = R2 = R$ in the schematic shown below.)

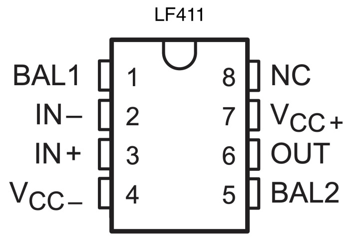
(c) Simulate the circuit using an LF411 op-amp. Read the **Notes** included in this document to learn more about finding this component. Using your simulation results, confirm that your design meets the spec. If you cut all the resistors exactly in half and double all the capacitors, it won't change the theoretical transfer function, but it will increase the effect of the output resistance. Try it. Does the transfer function still meet the spec? Find the lowest resistance for which the simulation still meets the spec. Ensure the gain does not go above -55 dB from 180 kHz to 1 MHz.



Save the simulated $|H(f)|^2$ as an ascii text file so that you can compare it with the theoretical filter using MATLAB. Read the **Notes** included in this document to learn more about this. Create a MATLAB script that will plot your theoretical $|H(f)|^2$, then read and plot your simulated $|H(f)|^2$ using distinct symbols (i.e., not connected by a line). Overlay two plots. If you want to be really fancy, have it plot the specs, too, as in the first figure in these instructions. Remember to include this script as well as the final plot containing the theoretical and simulated data sets in your final report.

3. **Experiment:** Build and test the active circuit. We will use the LF411 op-amp with the pins shown below. In our lab, we have lots of 1% resistors in stock, so you should be able to get pretty

close to the three resistors you need. You can always use a resister larger than the minimum value determined in your simulation to make the choice easier. However, we don't have nearly as wide a range of capacitors, you will probably have to put two capacitors in parallel to approximate C1 and the same for C2.



(a) Put probes on both the input and the output and trigger a sine waveform using a signal generator at the input probe. This will allow you to measure low-level signals at the output without losing the trigger. Measure the voltage gain at the spec frequencies: DC(use 5 or 10 Hz instead of 0 Hz), fPEAK, 4 kHz, 32 kHz, 180 kHz, and 1 Mhz. Create a table to report your measured input and output peak-to-peak voltage amplitudes and the calculated voltage gain in dB. Compare this with your theoretical and simulated values and comment on any discrepancies.

At high frequencies, the output voltage will be weak and noisy. You can't use the scope's peak-to-peak measurement with noisy data because it will give you the sum of the signal and noise. You will have to work to get the noise down:

i. use a strong signal as high as you can achieve on the signal generator (aim for at least 10 V_{pp}).

ii. use the probe in X1 mode (remember to change the channel setting to match the probe

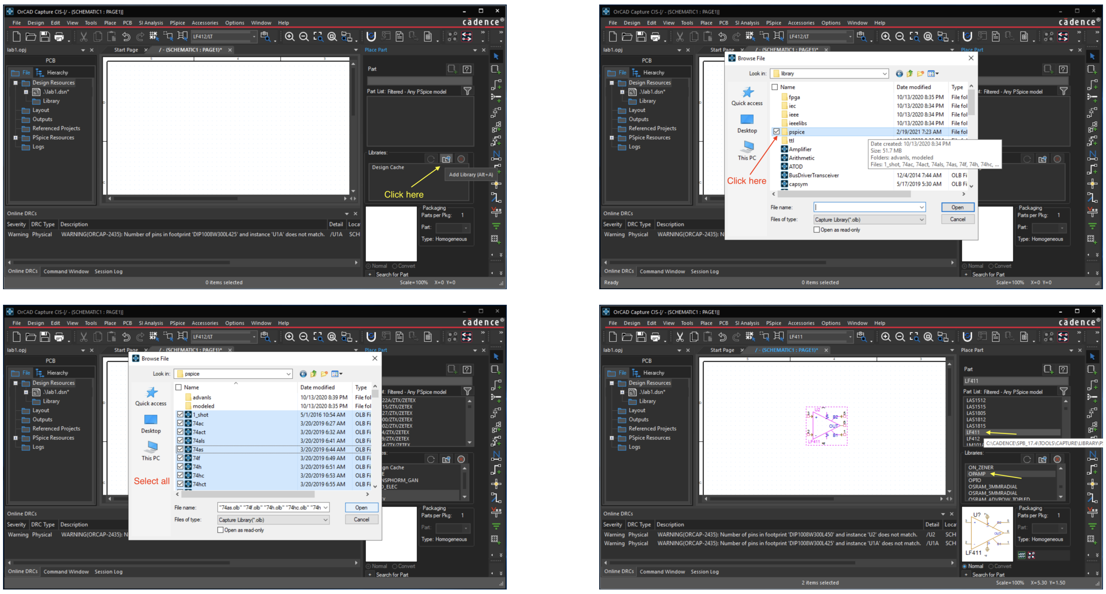
iii. use trace averaging

With all these features, you should be able to reach −60 dB. In this circuit, you can use X1 mode on the probe. This is often not possible because you need a high-impedance probe. But in this circuit, the probe is connected to the low impedance output of an op-amp, so the probe impedance is not a problem.

(b) Explore the remainder of the frequency range up to the generator limit and ensure there are no unexpected "features." Since this is an active circuit, you must ensure that the signal does not drive the op-amp into a nonlinear operating condition. So long as the output voltage still looks like a sine wave, you are probably OK. A good way to confirm that the system is being tested in a linear range is to change the input voltage by a factor of two and re-measure $|H(f)|^2$; if it is the same, you are OK. Take a scope image before and after changing the input voltage. If they differ, reduce the input voltage by another factor of two and try again. Put your measurements in an ascii file so you can read them into MATLAB and plot them on top of the simulation.

Notes:

Note 1: To add LF411 op-amp to your schematic on PSpice, add all the files in the PSpice library to your current library. LF411 op-amp is in PSpice/opamp library. Use the following pictures as a guide.



Note 2: To save a waveform data on PSpice, select the trace name (under the plot), then from the edit menu, choose copy. You can also right-click on the curve you want to copy. Open Notepad and paste the data in. You will see a header line and frequency followed by the amplitudes in dB. Save this with a .txt extension in a convenient place. To save a waveform data on LTSpice, select the trace name (above the plot), go to file, and click export data as txt. Then, select the waveform and click ok. It does not matter if you select Cartesian or polar coordinates"

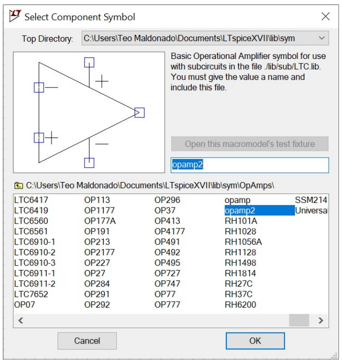
Note 3: To add LF411 op-amp to your schematic on LTspice, first download the LF411C PSpice Model (LF411C.text) file from the internet. The file includes the spice model for the LF411C op-amp. Save the file in your desired folder. If the file extension is not *.txt*, change the extension to *.txt*. Next, create a spice directive to include the LF411 library file. For example, if you use a Windows computer and your username is *Max* and you saved the file in the *ECE100labs* folder on your desktop, you should use the below spice directive.

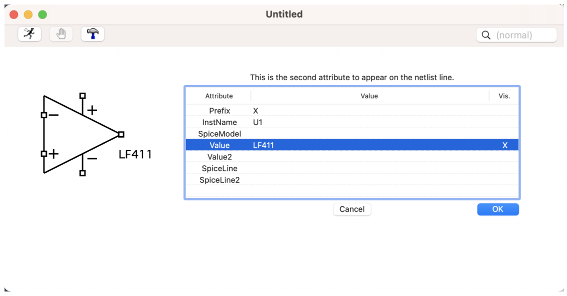
.lib c:\Users\Max\Desktop\ECE100labs\LF411C.txt

If you are a MAC user with username *Max* and saved the file on your desktop, you should use the below spice directive.

.lib /Users/Max/Desktop/LF411C.txt

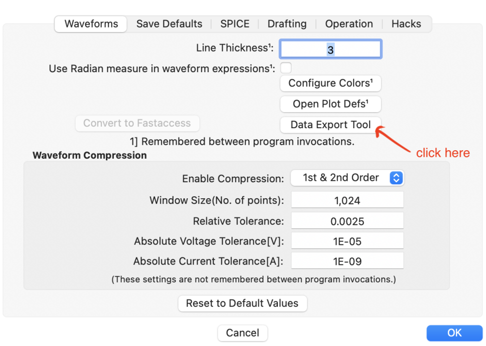
Place an "opamp2" component on your schematic. Once placed, right-click the component, verify it contains the prefix "X", and change its value to "LF411C". Use the following pictures as a guide.

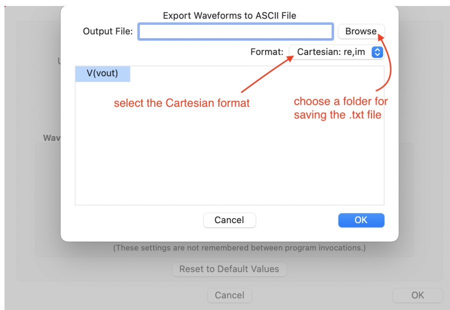




Notes 4: To save a waveform data in LTspice on Windows, while viewing the waveform tile (the waveform workspace), go to "File" then "Export data as text." A window will pop up where you will select V_{out}, choose where you would like to save the .txt file, then select "OK."

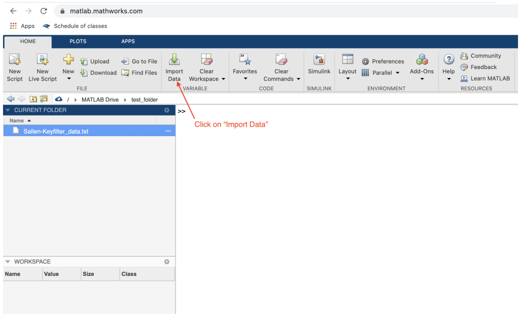
Notes 5: To save a waveform data in LTspice on MAC, while you are in the waveform viewing window, click on the Control Panel (the hammer icon), then click on "Waveforms", then click on "Data Export tool". Select a folder to save the output text file, then select the desired waveform to be exported. Choose the Cartesian format. You can use the following pictures for more guidance.

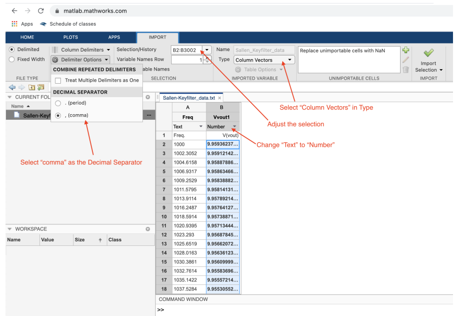


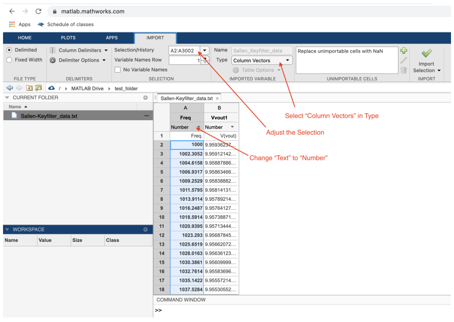


Notes 6: To import the LTspice data saved in the text file into MATLAB, open MATLAB,

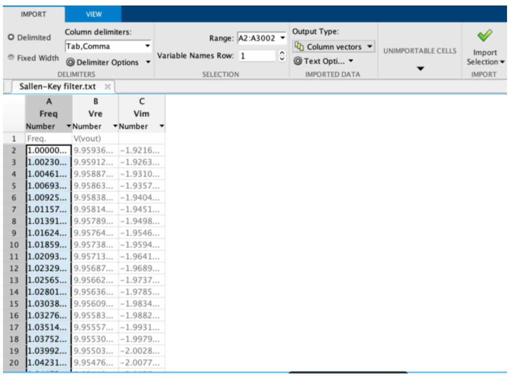
then click on "Import Data" and select your text file from the corresponding folder. Import the frequency values and the voltage values separately in two steps. You can use the following pictures for more guidance. MATLAB Online is used in the following three pictures.

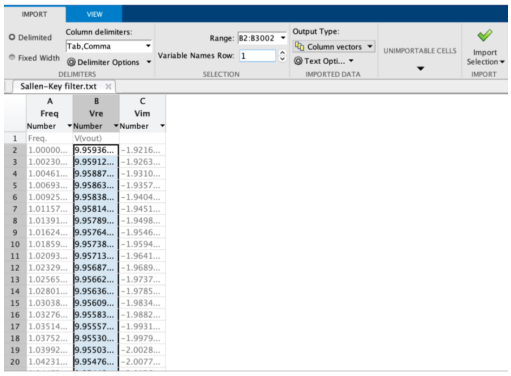


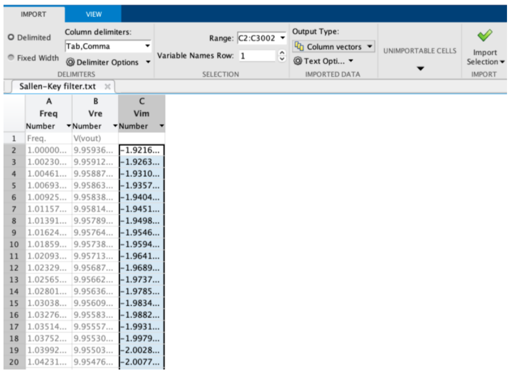




If you use the desktop version of MATLAB, apply the settings shown in the below pictures.







Report:

Please follow the provided lab report format. Your report should contain a short discussion of the simulations with the requested plots. It should include a diagram of the circuits with the actual components used. The most important part of the report is the plot showing that the transfer function meets the spec.

Make sure to include the following in your report:

- Hand calculations and the requested plots from 1a-d
- Hand calculations for 2a-b
- Plot from simulation using minimum working resistance value from 2c
- Script for plotting the theoretical $|H(f)|^2$, saving and plotting the simulated $|H(f)|^2$
- The final plot, overlaying the plots of theoretical, simulated, and measured $|H(f)|^2$. Comment on discrepancies.

This list is not all-encompassing, check with your TAs, but serves as a helpful checklist.