# FILTERS – Wk 12 (April 7th and April 9th 2020)

**LAB 12: SIMULATING THE FREQUENCY DEPENDENCE OF LOW PASS, HIGH PASS, AND BAND PASS FILTERS.**

## GOALS

In this lab, you will simulate the frequency dependence of the passive filters built in Lab 3. You will also gain experience in simulating effect of measurement tools.

- Learn how to perform AC analysis of circuits using LTspice
- Learn how to perform time domain analysis of the filters using LTspice

## DEFINITIONS (from the manual of Lab 3)

**Pass band** – the range of frequencies that can pass through a filter without being attenuated.

**Attenuation band -** the range of frequencies that a filter attenuates the signal.

**Cutoff frequency (or corner frequency or 3 dB frequency), fc** – the frequency boundary between a pass band and an attenuation band. **fc** is the frequency at the half-power point or 3dB point, where the power transmitted is half the maximum power transmitted in the pass band. The output voltage amplitude at f = fc is $1/\sqrt{2}$ = 70.7% of the maximum amplitude.

**Low pass filter** – a filter that passes low-frequency signals and attenuates (reduces the amplitude of) signals with frequencies higher than the cutoff frequency

**High pass filter** – a filter that passes high-frequency signals and attenuates (reduces the amplitude of) signals with frequencies lower than the cutoff frequency

**Band pass filter** – a device that passes frequencies within a certain range and rejects (attenuates) frequencies outside that range.

**Bandwidth of Band pass filter** – the range of frequencies between the upper ($f_+$) and lower ($f_-$) half power (3dB) points: bandwidth $\Delta f = f_+ - f_-$.

## FILTER BASICS (from the manual of Lab 3)

The 3 dB frequency of RC Low- and High-pass filters is

$$f_c = \frac{1}{2\pi RC},$$

where fc ( or $f_{3dB}$)is the 3 dB or half-power point.

**The resonance frequency of and characteristic impedance of a parallel LCR band-pass filter**

$$f_0 = \frac{1}{2\pi\sqrt{LC}} \qquad Q = \omega_0 RC = \frac{f_0}{\Delta f}$$

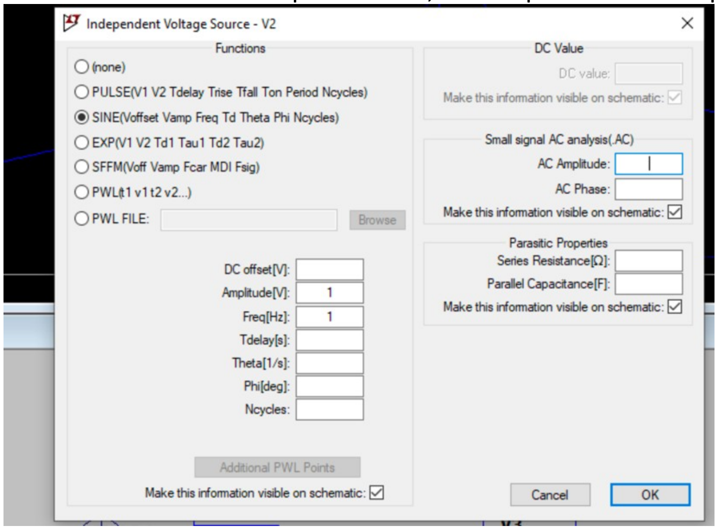
where $\omega_0 = 2\pi f_0$. The resonant frequency, $f_0$, is the center frequency of the pass band, and the Q is equal to the ratio of the center frequency to the bandwidth Δf. (These definitions are exactly true only if Q>>1).

## LAB PREP ACTIVITIES

**Activity 1: Transient Simulation of LTspice**

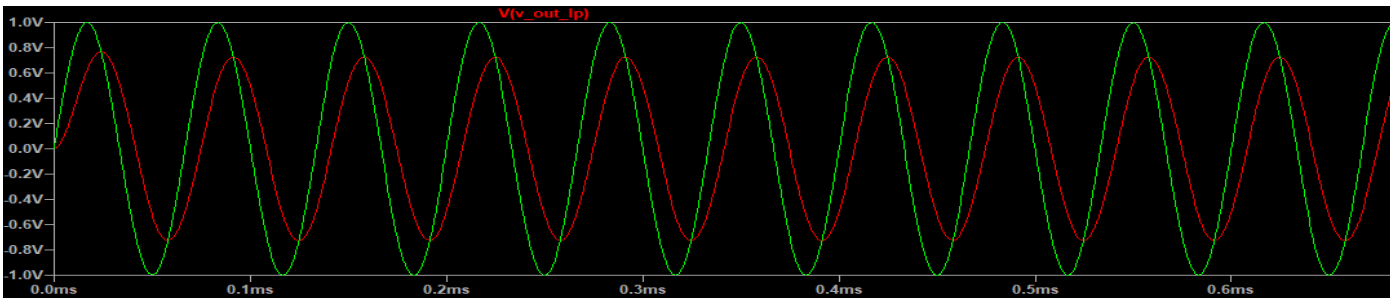
Transient method: In this method, which simulates the way we found the frequency response in Lab 3, we set the power source to a function, e. g. SINE Function, and simulate the circuit in Transient mode at a frequency. The result is various output such as voltage as a function of time. This method is similar as displaying the signal on an oscilloscope. Below are the steps to run Transient Simulation.

a) Open Voltage Source Editor and select a function and its parameters. In this case SINE function is selected and its parameters, the amplitude and frequency are typed in.




b) Open Edit Simulation, choose Transient, and set stop time considering the frequency of the function, typically several period of the functions.

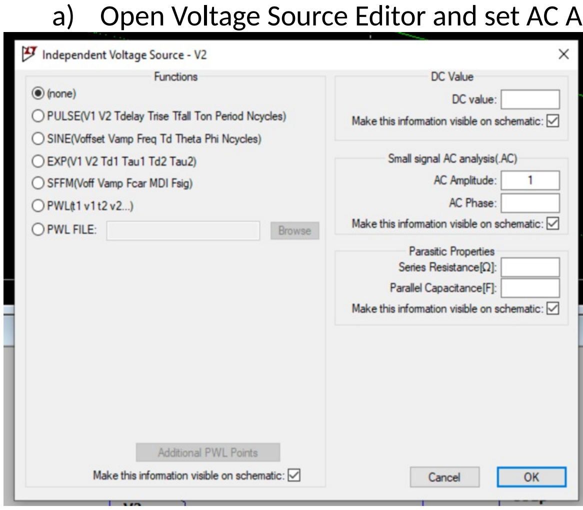
c) An example of a Transient simulation is shown below. The input is in green and the output, in red.



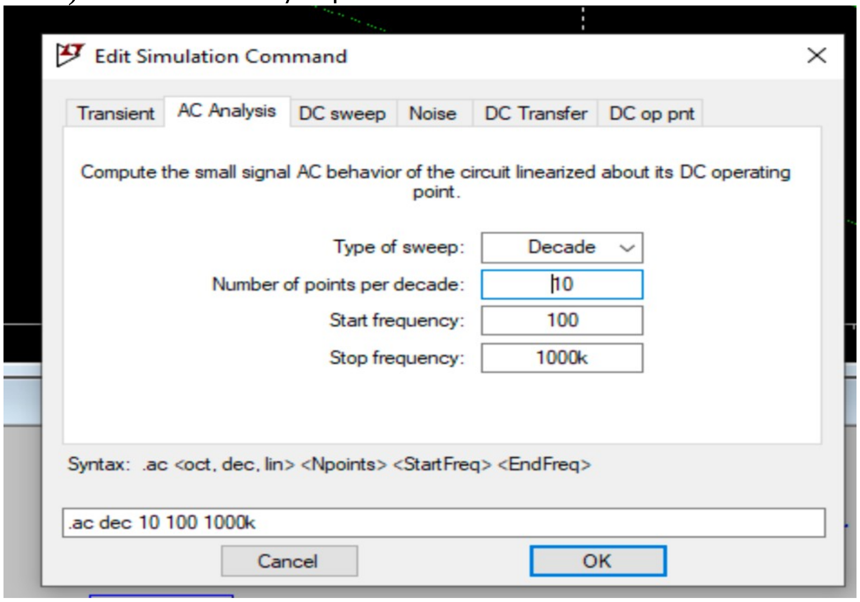

**Activity 2: AC Analysis Simulation of LTspice**

AC Analysis: In this method, simulation is done in AC Analysis mode with a set AC amplitude of the 'Small signal AC analysis' of Voltage Source. Notice that the result of simulation is a bode plot and y-axis represent output power relative to the input power in the unit of dB.

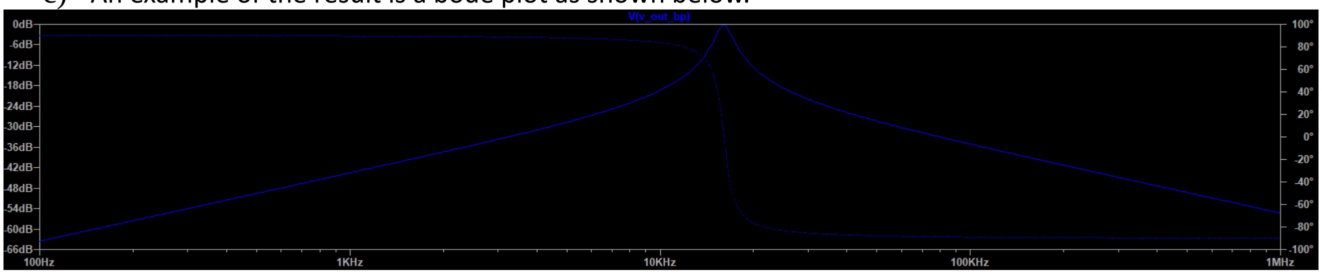
a) Open Voltage Source Editor and set AC Amplitude of Small Signal Analysis, e. g. 1 V.



b) Set the AC Analysis parameters of the Edit Simulation Command. See an example below.



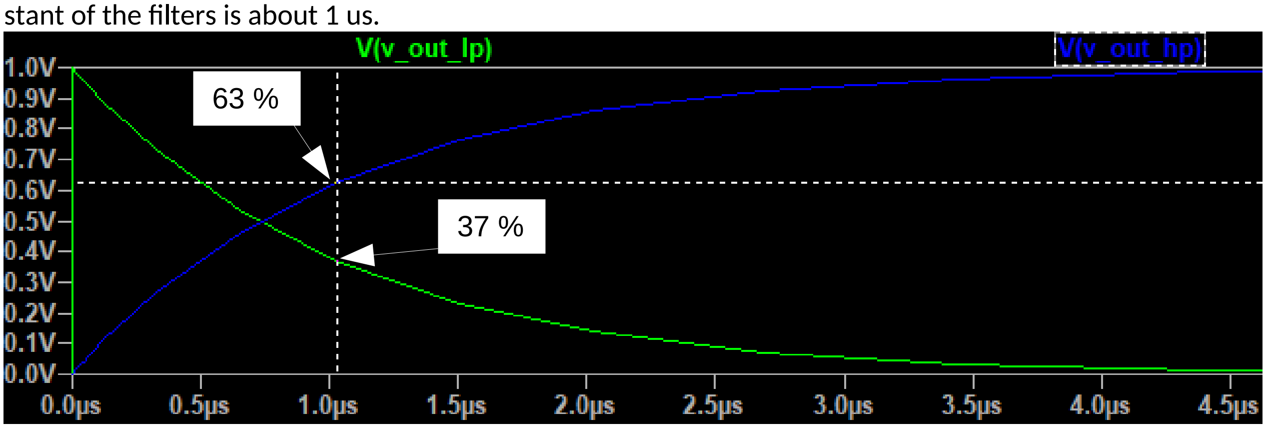

c) An example of the result is a bode plot as shown below.




d) Refer to the tutorial link below for more details of AC analysis.
https://www.youtube.com/watch?v=fziUQaVQxA4

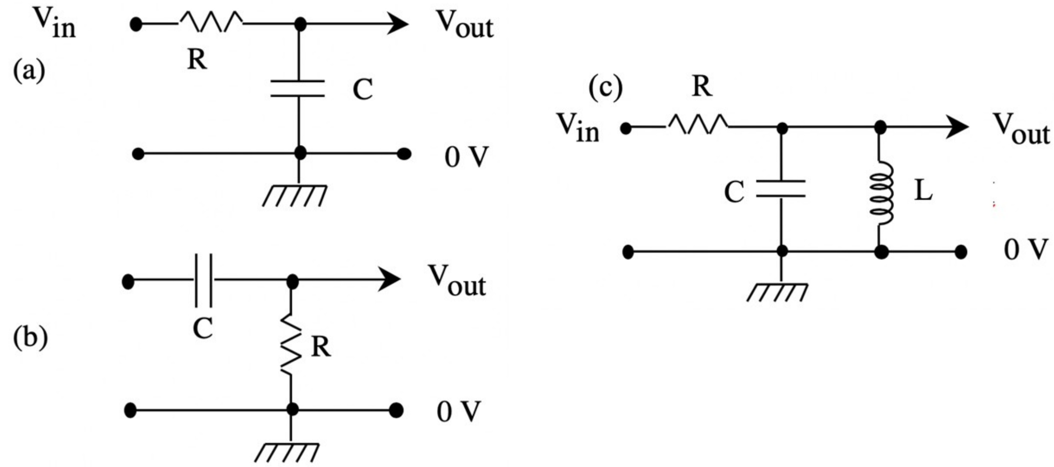
**Activity 3: Finding time constants of the filters**

The transient behaviors of the filters shown below, are characterized by time constants of the circuits, R x C and L/R, for RC and RL filters, respectively. The characteristic time of the low-pass filter can be found by measuring the time it takes for the output voltage reach 63% (or 1-1/e) of an input step voltage after it is applied. The characteristic time of a high-pass filter can be found by measuring the time it takes for the output voltage drops to 37% (or 1/e) of an initial value after an input voltage is applied. The graph bellow shows output of an RC low-pass (blue) and a RC high-pass (green) filters when a step function is applied at time zero. The time constant of the filters is about 1 us.



SETTING UP THE CIRCUITS AND SIMULATING THE BEHAVIOR

Figure 1. Circuits used in the lab (a) Low-pass filter (b) High-pass filter (c) Band-pass filter.




**Step 1: Building the Circuits**

a) Open a schematic of LTspice and gather all the components to be able to build the four circuits shown in Fig. 1. Type in **the values of your components measured in your Lab 3**. The values of the components should be close to the values as below.
   1. Low-pass filter: R = 10 kΩ, C = 1000 pF
   2. High-pass filter: R = 10 kΩ, C = 1000 pF
   3. Band-pass filter: R = 10 kΩ, C = .01 µF, L = 10 mH

b) Build all three circuits using LTspice on separate windows. Calculate the time constant and the 3dB frequency using the filter equations.

**Step 2: Test the circuits and different wave form**

a) It is helpful to simulate both the input voltage as well as the output voltage at the same time. Simulate output signal of each circuit with input signal of 1 kHz sine wave with 1 volt p-p using **Transient method** (See Lab Preb Activity 1). Confirm the waveform frequency and amplitude of the input and output.

b) How to make **square wave**? Switch your Vsource to pulse and specify the following values (See Fig. 2) to yield 1kHz square wave, starting 0V for 0.5ms at the very beginning!
Note that you should adjust **Tperiod** in order to change the period**.** **Trise** and **Tfall** should be much smaller than Tperiod. If you want to have symmetric, i.e. Vlow = -2.5V and Vhigh = 2.5V, (instead of 0 and 5 V), you can change **Vinitial** and **Von** to those values.

c) Now, let's practice how to make **triangular wave** with f = 1kHz, Vpp = 5 V .
Vintial : -2.5 (or 0 V, which the minimum voltage of your triangular wave is 0 V, instead of -2.5V)
Von : +2.5
Tdelay : 0 (this determines where the triangular wave starts)
Trise : 0.5ms (This MUST be **Tperiod/2** for triangular wave)!
Tfall : 0.5 m
Ton : 0
Tperiod : 1m
Ncycles: [leave blank – means N = infinity]

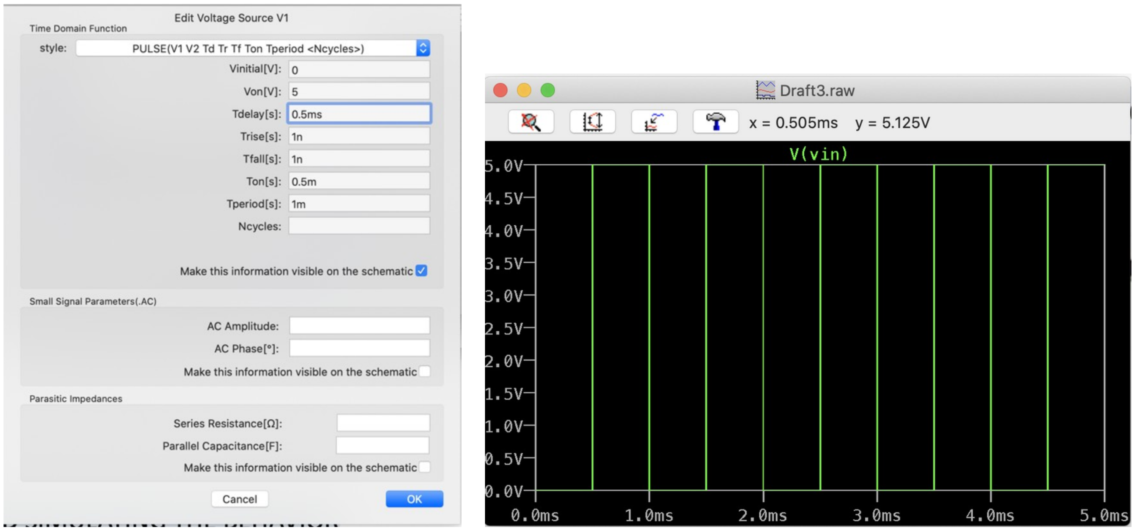


Figure.2 Screenshot to set up the 1kHz square wave, starting at 0V(low) and 5V (high)

**Step 3: Simulate the frequency dependence of the low- and high-pass filters.**

a) Simulate the gain of the low-pass filter (Fig. 1(a)) over a range in frequency (100 Hz to 1 MHz) using the **AC Analysis method** (See Lab Preb Activity 2). Determine the cut-off frequency for the low-pass filter. Compare your simulated half power point (3dB point or Vout/Vin = 0.707) with the cut-off frequency computed and measured in Lab 3.

b) Repeat Step 3 a) with the high-pass filter (Fig. 1(b).

**Step 4: Simulate the frequency dependence of the band-pass filter.**

a) Simulate the band-pass filter (Fig. 1(c)) using the **AC analysis method**. Find the resonance frequency. Find the transfer function and the phase relative to the input signal at the resonance frequency. On resonance, Vout will be a maximum and the phase shift between the input and output waveforms will be zero.

b) Determine the quality factor Q by finding the frequencies at the two half-power points $f_+$ and $f_-$ above and below the resonance at $f_0$ from the simulated bode plot in part a). Recall that

$$Q = \frac{\text{Resonant frequency } f_0}{\text{Bandwidth } \Delta f} \qquad \text{where } \Delta f = f_+ - f_- .$$

c) Compare the simulated value of Q with that predicted and the measured in Lab 3. Do they agree, respectively?

**Step 5: Simulate the transient behavior of the low/high pass filters, using Square and Angular Waveforms.**

a) Simulate the time domain response of the low-pass filter using the **Transient Time method** by selecting Pulse Function in the Voltage Source. Set the parameters of the pulse as follows, Vintial = 0, Von = 1 V, Tdelay = 0, Trise = 1ps, Tfall = 1ps, Ton = 100us, Tperiod = 200us. Run the Transient Time simulation for the following cases and briefly discuss the behavior.

| Simulation Cases | Ton | Tperiod | Simulation Time |
|---|---|---|---|
| Case 1 | 100 us | 200 us | 1 ms |
| Case 2 | 20 us | 40 us | 200 us |
| Case 3 | 5 us | 10 us | 50 us |

b) Repeat part a) with the high-pass filter.

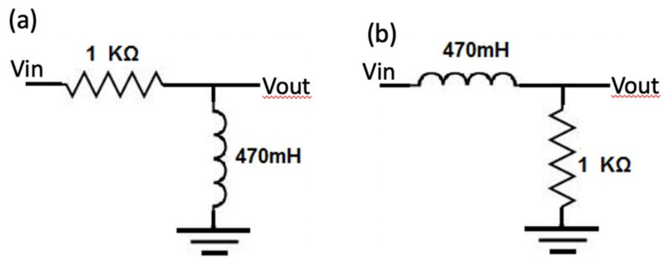
c) Pick 3 suitable frequencies of square wave , $f_1$, $f_2$ and $f_3$, where $f_1 << f_{3dB} \approx f_2 << f_3$ and run transientanalysis. For example, $f_1 = f_{3dB}/10$, $f_2 = 1.1*f_{3dB}$ and $f_3 = 10*f_{3dB}$.
Check at what frequency range, these filter circuits acts like differentiator or integrator.
Simulation time (***Stop time***) should be picked to include at least 3-4 period of signals.

d) Repeat **Step 5a)- c)** with **Triangular wave** – note that for **Triangular wave, the setting should be** Trise=Tfall=Period/2 and Ton = 1ps (≈ 0). *Please include your screen shots of* Vin and Vout for three different frequencies ($f_1$, $f_2$, $f_3$*) in your write-up.*

e) Compare the Vout wave forms for three frequencies whether high/low pass filter acts like integrator or differentiator. In order to work properly as integrator/differentiator, what frequency should be used ?

**[Extra credit 10 pts] Step 6: Simulate the transient behavior of the low-pass and high- filters.**

a) Let's build RL filter as shown below using LTspice. Check their frequency responses to tell which one is high or low pass filter?




b) What is your measured value for the 3dB frequency of both filter circuits above ?
(Hint: According Kirchoff's law, the differential equation for this circuit and solution

$$L\frac{di(t)}{dt} + Ri(t) = V_{\text{in}}(t)$$

, where

$$i(t) = \frac{V_0}{R}(1 - e^{-t/(R/L)})$$

$$V_R = Ri(t), \quad V_L = L\frac{di}{dt}$$

.

Thus,

$$V_R = V_0(1 - e^{-t/(R/L)}); \quad V_L = V_0 e^{-t/(R/L)}$$

.

c) Pick 3 suitable frequencies, $f_1$, $f_2$ and $f_3$, where $f_1 << f_{3dB} \approx f_2 << f_3$ and run transient analysis for the square wave as **Step 5a)- c).** Can both RL filter circuit be used as differentiator and integrator? (*Please include your screen shots for three different frequencies Vin and Vout in your write-up).*

**Appendix: Number convention of LTspice (Modified from Wikipedia)**

| SPICE Suffix[26] | Metric Name | English Name | Power of 10 | Numeric Value | Notes and Common Mistakes |
|---|---|---|---|---|---|
| u or µ | micro | Millionth | $10^{-6}$ | 0.000001 | Older SPICE software does not support the µ (Mu) character |
| T | tera | Trillion | $10^{12}$ | 1000000000000 | |
| p | pico | Trillionth | $10^{-12}$ | 0.000000000001 | |
| n | nano | Billionth | $10^{-9}$ | 0.000000001 | |
| mil | thou | | $25.4 \times 10^{-6}$ | 0.0000254 | mil is a thousandth of an inch (0.001") which is 25.4 µm[26] |
| MEG | mega | Million | $10^{6}$ | 1000000 | Wrong use of m/meg/mil are common mistakes in all SPICE programs |
| m | milli | Thousandth | $10^{-3}$ | 0.001 | "1m" & "1M" doesn't mean "1megaohm, instead "1MEG" is correct[1] |
| K | kilo | Thousand | $10^{3}$ | 1000 | |
| G | giga | Billion | $10^{9}$ | 1000000000 | |

- The suffix (left column) is case insensitive.[1] For example, 1MEG / 1meg / 1Meg represents 1000000; 1k / 1K represents 1000.
- Any appended text after the suffix (left column) is ignored.[1] For example, 2MegHz / 2MegaOhm represents 2000000; 3mV / 3mOhm represents 0.003.
- In LTspice, any suffix (left column) can replace the decimal point of a real number, a common format for printed schematics.[1] For example, 4K7 represents 4700, 1u8 represents 0.0000018.

[1] LTspice - General Conventions". *Ltwiki*. Archived from the original on December 5, 2018.