ECE 214 - Lab #3 — Filter Design

10 February 2020

Introduction: In this lab, you will design, simulate, build, and test a low-pass filter. The block diagram of the filter is shown in Figure 1.

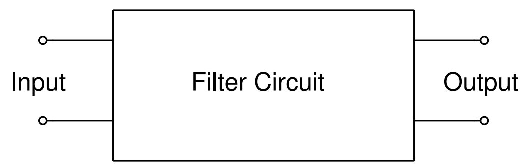

Figure 1: Block diagram of the filter circuit.

The filter circuit should satisfy the following specification:

1. Input: Square wave with a frequency of 1.25 Hkz and a peak-to-peak voltage of 4 V.
2. Output: Peak-to-peak voltage greater than 2 V, and magnitude of the third harmonic at least -20 dB below the magnitude of the fundamental frequency.

These specifications are illustrated by the signals in Figure 2.

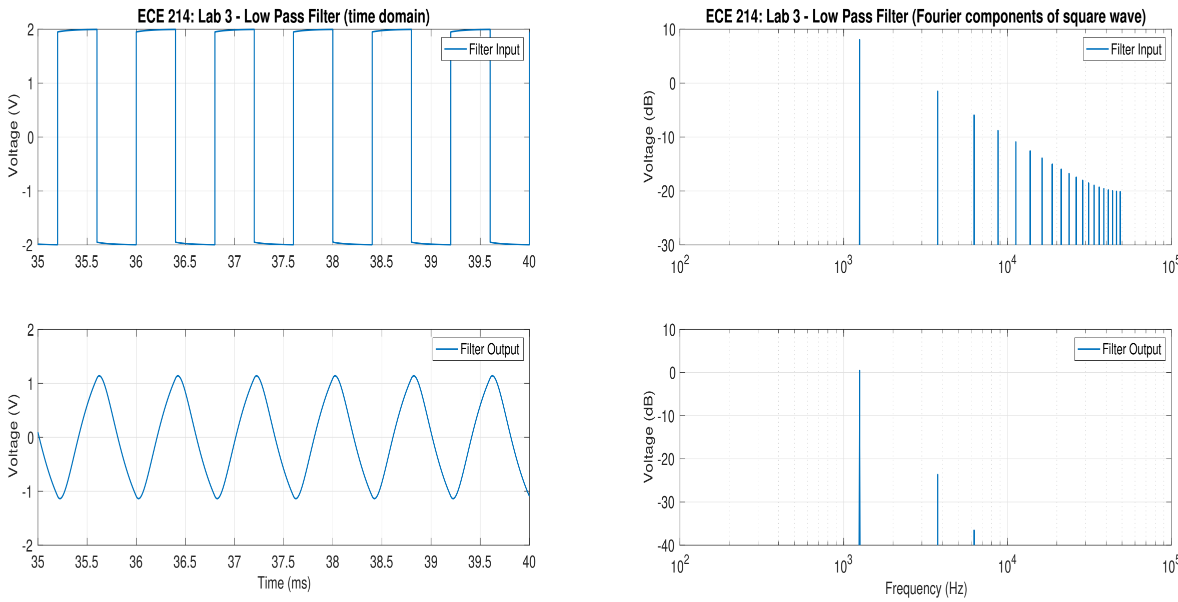

Figure 2: Input and output signals from the filter in the time domain (left) and the frequency domain (right).

Pre-Lab:

1. Design a low-pass filter to satisfy the above specification.
2. Simulate the filter response in the time domain, and verify the design satisfies the specification.
   (a) Plot the voltage at the input of the filter, and the output of the filter, as a function of time.
   (b) Use the MATLAB® function "time_to_freq," available on the course website, to approximate the Fourier series of the time-domain signals. Plot the voltage at the input of the filter, and the output of the filter, as a function of frequency. Show the voltages in dB.

Lab Procedure:

1. Build the filter you designed in the Pre-Lab.
2. Connect the FG to the input of the filter. Set the FG to produce a 4 V peak-to-peak square wave signal at a frequency of 1.2 kHz. Measure the input and output of the filter on the scope in both the time-domain and the frequency-domain. Record the time- and frequency-domain signals in your notebook. Use a USB stick to download images from the oscilloscope screen. Make sure all axes and peaks are properly labeled.
3. Does the output of the filter meet the specifications?
   (a) If "Yes:" you are done with the Lab Procedure.
   (b) If "No:" repeat this lab starting at step 1 in the Pre-Lab.

Post-Lab:

1. Compare the measured results with the simulated results. Make a note of any discrepancies between the measurements and simulations.
2. In addition to the transient simulation, useful information on the response of a filter can be obtained using AC, or small signal, simulations. AC simulations are similar to Phasor analysis. During AC simulations, the circuit response is simulated as a function of frequency. To learn more about AC and transient simulations, you should read Sections 1.2.2, 15.3.1 and 15.3.9 of the NGspice user manual `http://ngspice.sourceforge.net/docs/ngspice-manual.pdf`.
3. Simulate the frequency response of the circuit using AC simulation. (Uncomment the last sections of the MATLAB® file for this lab to enable the AC simulation.)
   (a) Set the AC Voltage of the pulse generator in your schematic to 1V. The AC Voltage is the magnitude of the voltage used when performing AC simulations.
   (b) For AC simulations, the frequency variable in the NGspice data file is called `FREQUENCY`.
   (c) By adding the `.ac dec 201 1e2 1e5` control statement in your .m file, the simulation will perform a frequency sweep containing of 201 frequency values ranging from 100 Hz to 100,000 Hz using a logarithmic scale.

(d) Plot the magnitude of the output voltage `Vout` in decibels as a function of frequency. To convert the output voltage to dB, use: `20.*log10(abs(Vout))`.

(e) Plot the phase of the output voltage `Vout` in degrees as a function of frequency. To convert the output voltage to phase, use: `phase(Vout).*180/pi`.

4. Compare the results from the time domain simulations in Pre-Lab Section #2, to the results of the AC simulations in Post-Lab Section #3. Explain the relationship between these two types of simulations. Do the time domain and frequency domain simulation results provide the same information. Are the results from the two types of simulations consistent?