ECE 214 - Lab #3 — Filter Design

13 February 2018

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.

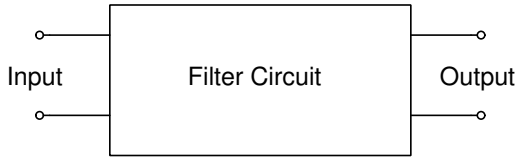

Figure 1: Block diagram of the filter circuit.

The filter circuit should satisfy the following specification:

1. Input: Square wave with a frequency of 800 Hz and a peak-to-peak voltage of 4 V.
2. Output: Peak-to-peak voltage greater than 2 V, and the third harmonic at least -20 dB below the fundamental frequency.

These specifications are illustrated in Figure 2.

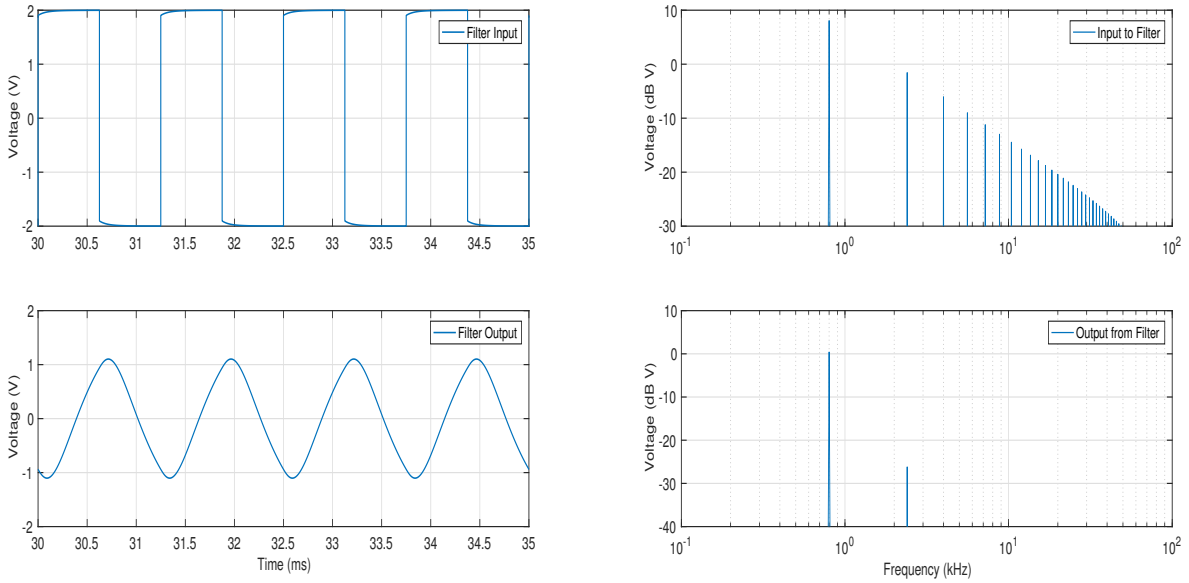

Figure 2: Input and output signals from the filter in the time domain (left) and the frequency domain (right).

Pre-Lab:

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

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 800 Hz. 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. Make sure all axes and peaks are properly labeled. Photographs of the scope screen are an efficient way to record this data.
3. Does the output of the filter meet the specifications?
   (a) If "Yes:" you are done with the Lab Procedure.
   (b) If "No:" repeat the lab starting at step 1 in the Pre-Lab.

Post-Lab:

1. Compare the measured results with the simulated results from NGspice. 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, and the circuit response is simulated as a function of frequency. You should look at Section 1.2.2 and Sections 15.3.1 and 15.3.9 of the NGspice user manual http://ngspice.sourceforge.net/docs/ngspice-manual.pdf to learn more about AC and transient simulations.
3. Use NGspice to analyze your filter using `ac` analysis. You will need to generate a new MATLAB® .m file to perform the AC simulation and read in the frequency and the output voltage.
   (a) Replace the pulse generator in your schematic with a sine wave generator and set the AC Voltage 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) Replace the `.tran` control statement in your .m file with `.ac dec 201 1e2 1e4`. This will perform a frequency sweep containing of 201 frequency values ranging from 100 Hz to 10,000 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`.