

ECE 214 - Lab #3 — Filter Design

7 February 2017

Introduction: In this lab you will design, simulate, build, and test a filter to alter the frequency characteristics of a square wave as shown in Figure 1 and Figure 2.

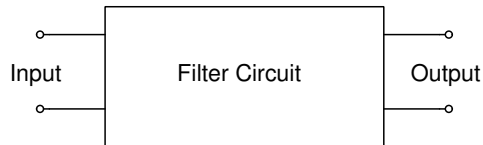


Figure 1: Block diagram of the filter circuit.

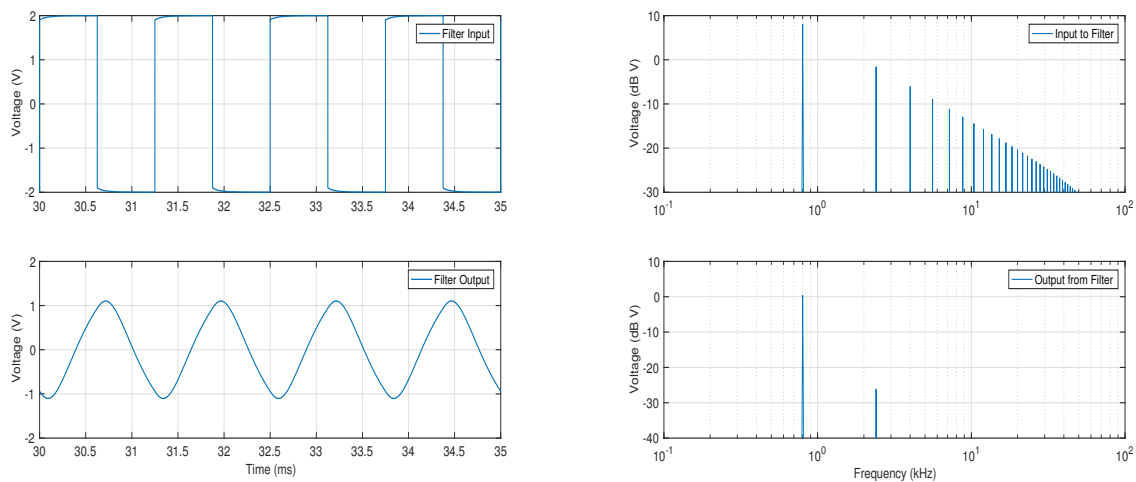


Figure 2: Input and output signals from the filter in both the time domain (left) and the frequency domain (right).

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; third harmonic at least -22 dB below the fundamental frequency.

Pre Lab:

1. Design a filter to satisfy the above specification.
2. Use NGSpice to simulate the filter and verify the design.
 - (a) Plot the voltages at the input and output of the filter in the time domain.
 - (b) Plot the voltages at the input and output of the filter in the frequency domain.

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. You should look at Section 1.2 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. User NGSpice to analyze your filter using **ac** analysis:
 - (a) Replace the pulse generator in your schematic with a sine wave generator and set the AC Voltage to 1V.
 - (b) Replace the **.tran** statement in your **hspc** file with **.ac dec 201 1e2 1e4**. This will perform a frequency sweep consisting of 201 frequency values from 100 to 10,000 Hz using a logarithmic scale.
 - (c) Create a new **.m** file to read in the frequency and the output voltage. The frequency variable in the NGSpice data file is **FREQUENCY**.
 - (d) Plot the magnitude of the output voltage in dB $20 \cdot \log_{10}(\text{abs}(\text{Vout}))$ as a function of frequency.
 - (e) Plot the phase of the output voltage in degrees $\text{phase}(\text{Vout}) \cdot 180/\pi$ as a function of frequency.