**ECE 214 - Lab #3 — Filter Design**

15 February 2021

## 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.

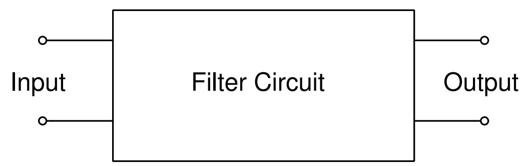


Figure 1: Block diagram of the filter circuit.

The filter circuit should satisfy the following specifications.

1. Input: Square wave with a frequency of 1.25 kHz and a peak-to-peak voltage of 4 V.
2. Magnitude of the input impedance: $|Z_{IN}| \geq 10$ k$\Omega$.
3. Output: Peak-to-peak voltage greater than 2 V and magnitude of the third harmonic at least -20 dB below the fundamental frequency.

These specifications are illustrated by the signals in Figure 2.

## Parts List

The values and number of resistors and capacitors are dependent on the design.

## Pre-Lab

1. Design a low-pass filter that satisfies the above specification.
2. Simulate the filter with a square wave input signal with a frequency of 1.25 kHz and a peak-to-peak voltage of 4 V, and verify that the signal at the output of the filter satisfies the specification. The MATLAB® file `ECE214_2021_Lab3.m` for NGspice is available at https://ece214.davidkotecki.com/docs/Matlab/ECE214_2021_Lab3.m. The MATLAB® function `vt_to_vf`, available at https://ece214.davidkotecki.com/docs/Matlab/vt_to_vf.m, is needed to approximate the Fourier series of the time-domain signals, and should be placed in the same directory as the `ECE214_2020_Lab3.m` file.
3. Plot the voltage at the input of the filter, and the output of the filter, as a function of time.
4. 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.

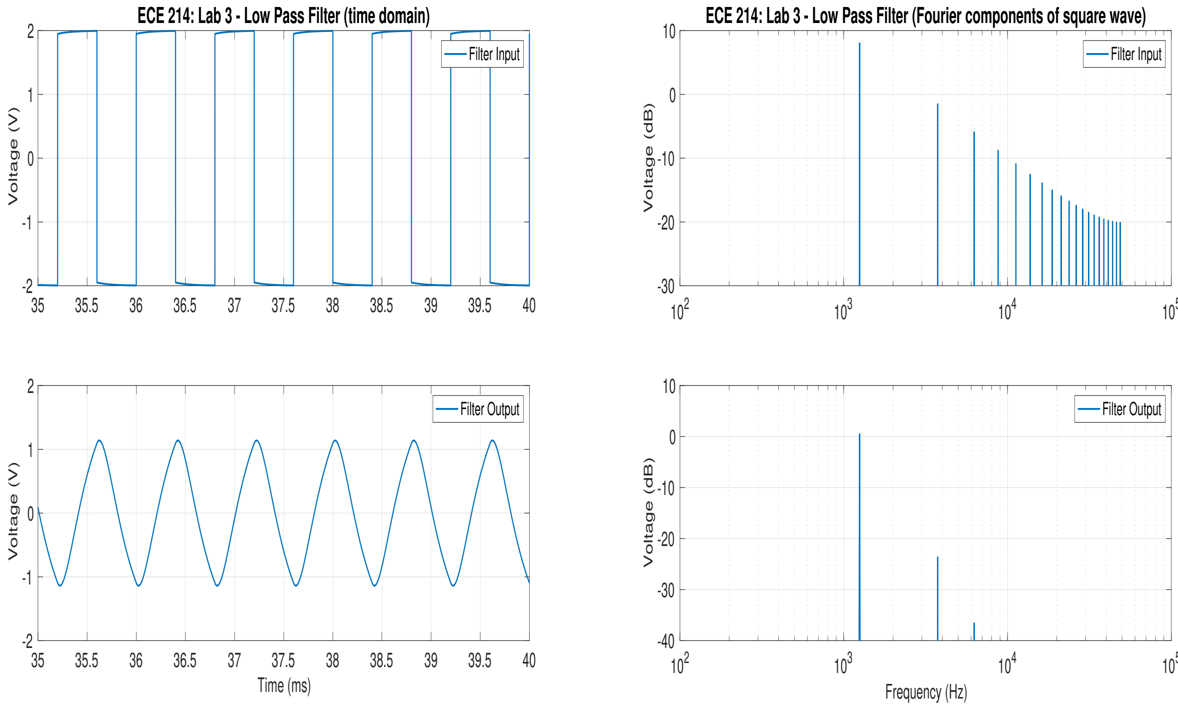


Figure 2: Low pass filter input signal (top) and output signal (bottom) in the time domain (left) and the frequency domain (right).

## Lab Procedure

1. Build the filter designed in the Pre-Lab.
2. Connect the WG to the input of the filter. Set the WG to produce a 4 V peak-to-peak square wave signal at a frequency of 1.25 kHz. Measure the input and output signals on the Analog Discovery 2 (AD2) in both the time-domain and the frequency-domain. Record the time- and frequency-domain signals in your notebook.
3. Does the output of the filter meet the specifications?
   (a) If “Yes” go to item 4 in the Lab Procedure.
   (b) If “No” repeat this lab starting at step 1 in the Pre-Lab.
4. The AD2 contains a Network Analyzer. The network analyzer drives a circuit with a swept sine wave and measures the circuit response as the input frequency changes. The output magnitude and phase can be displayed in a Bode format. Use the Network Analyzer in the AD2 to measure the frequency response of the filter.
   (a) Plot the measured magnitude of the output voltage `Vout` in decibels as a function of frequency.
   (b) Plot the measured phase of the output voltage `Vout` in degrees as a function of frequency.

**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 `ECE214_2021_Lab3.m` MATLAB® file to enable the AC simulation.)

   (a) Set the AC Voltage of the pulse generator in your schematic to 1 V. 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: `angle(Vout).*180/pi`.

4. Compare the time domain simulations from Pre-Lab item 2, the measured frequency response from Lab Procedure item 4, and the AC (frequency response) simulations from Post-Lab item 3.

   (a) Explain the relationship between the time domain simulations from Pre-Lab item 2 and the AC (frequency response) simulations from Post-Lab item 3.

   (b) Do the time domain and frequency domain simulation provide the same information?

   (c) Are the results from the two simulations consistent?

   (d) Plot the AC (frequency response) simulations from Post-Lab item 3 and the measured frequency response from Lab Procedure item 4 on the same graph. Compare the simulated frequency response to the measured frequency response.