ECE 214 - Lab #3 — Filter Design

7 February 2022

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.
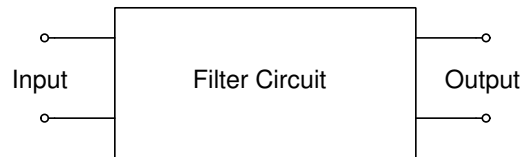

Figure 1: Block diagram of the filter circuit.

Specifications

1. Input: Square wave with a frequency of 1.25 kHz and a peak-to-peak voltage of 1 V.

2. Magnitude of the input impedance: $|Z_{IN}| \geq 10$ kΩ.

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 shown in Figure 2.

Parts List

The resistor and capacitor values are dependent on the design. A list of resistor and capacitor values in the 224 Barrows parts cabinet is located at: https://ece214.davidkotecki.com/docs/ECE214_Parts.pdf.

Pre-Lab

1. Design a low-pass filter that satisfies the above specification.

2. Simulate the filter response using a square wave at a frequency of 1.25 kHz and a peak-to-peak voltage of 1 V. Verify that the signal at the output of the filter satisfies the specifications listed above. The MATLAB® file ECE214_2022_Lab3.m for NGspice is available at https://ece214.davidkotecki.com/docs/Matlab/ECE214_2022_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_2022_Lab3.m file.

3. Plot the voltage, in volts, at the input and output of the filter as a function of time.

4. Plot the voltage, in dB, at the input and output of the filter as a function of frequency.
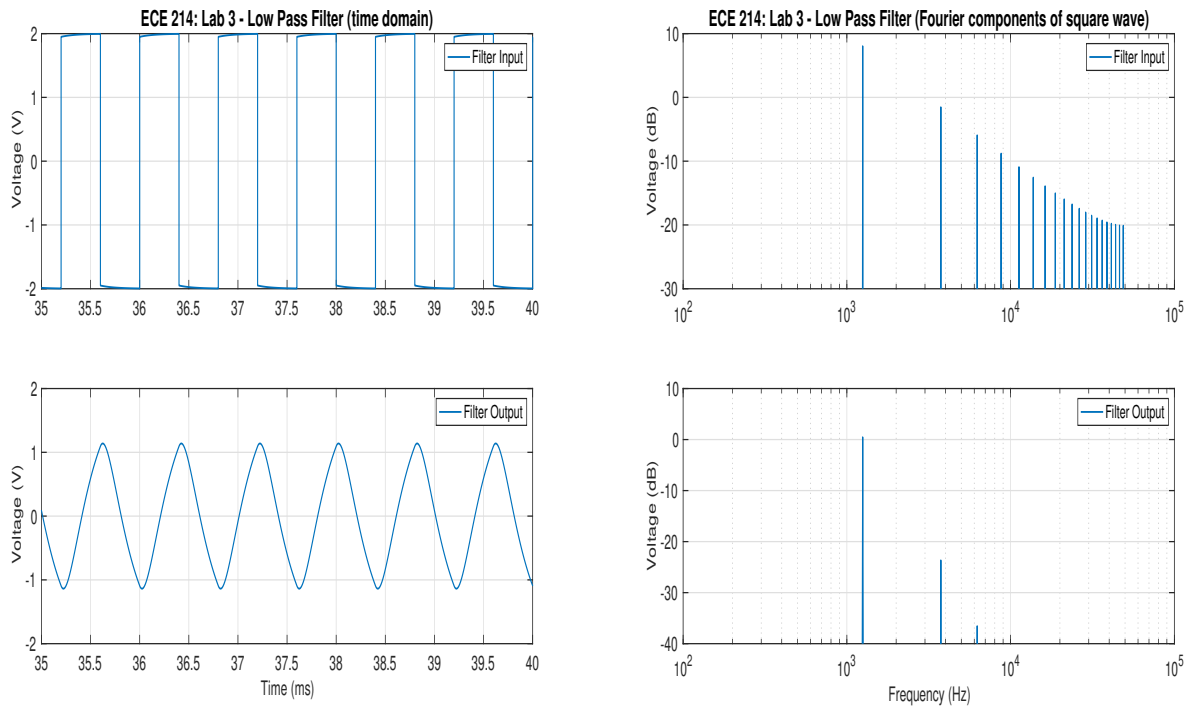
Figure 2: Low pass filter input signal (top) and output signal (bottom) in the time domain (left) and the frequency domain (right).

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 1 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 frequency swept sine wave and measures the circuit response as the input frequency changes. The output magnitude and phase are displayed as a function of frequency. 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 performing a 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 'return' statement at line 130 in 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 AC (frequency response) simulations from Post-Lab item 3, and the measured frequency response from Lab Procedure item 4.

   (a) Explain the relationship between the simulations from Pre-Lab item 2 and the simulations from Post-Lab item 3.

   (b) Are the results from these two simulations consistent?

   (c) 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. (Uncomment and edit lines 132-136, 159-161, and 170-172, in the `ECE214_2021_Lab3.m` MATLAB® file.) Compare the simulated frequency response to the measured frequency response.