ECE 214 - Lab #3 — Filter Design

14 February 2023

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.

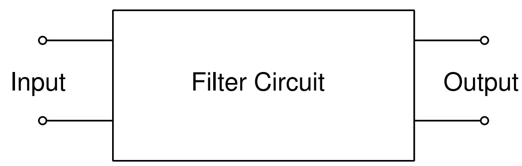

Figure 1: Block diagram of the filter circuit.

Specifications

1. Input: Square wave with a frequency of 1.25 kHz and a peak-to-peak voltage of 4 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 4 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.

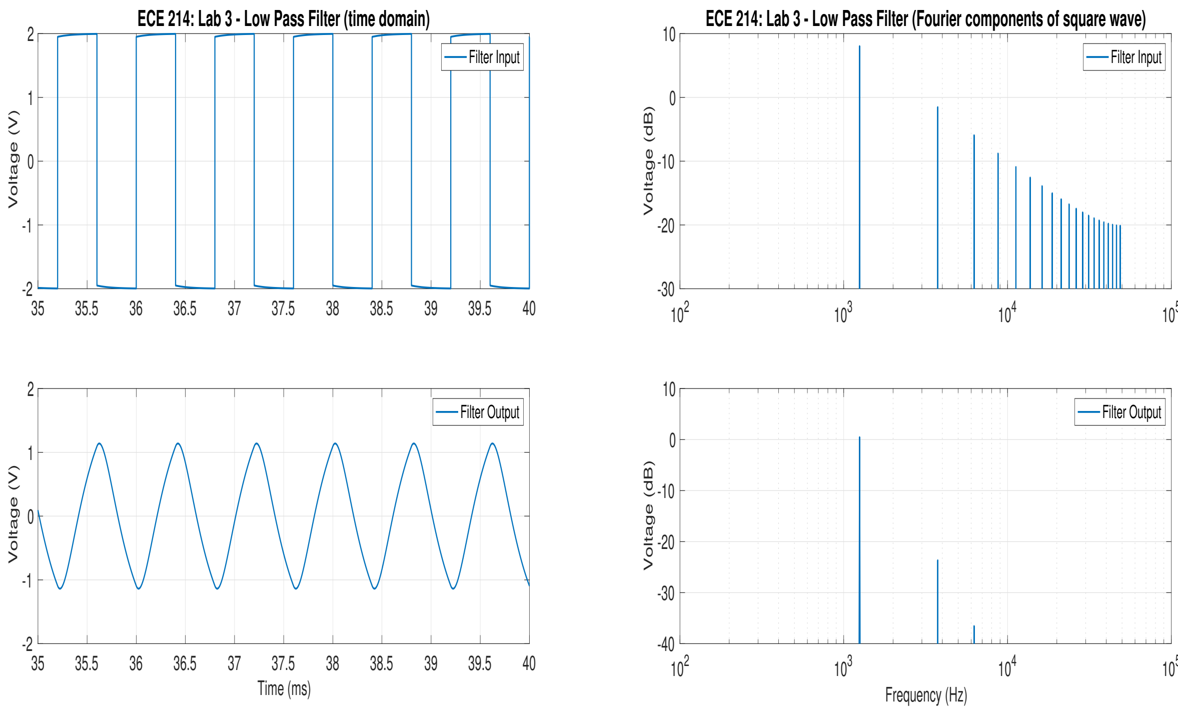

Figure 2: Low pass filter input signal (top) and output signal (bottom) in the time domain (left) and the frequency domain (right).

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 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 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. Save the measured magnitude and phase of the frequency response as a CSV file. This data will be imported into MATLAB® .

Post-Lab

1. Compare the measured time- and frequency-domain 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. During AC simulation, the circuit response is simulated as a function of frequency; AC simulations provide similar results to that obtained from the Network Analyzer. 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. (Comment the 'return' statement at line 129 in the `ECE214_2022_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) Compare the AC (frequency response) simulations from Post-Lab item 3 and the measured frequency response from Lab Procedure item 4. (Uncomment and edit lines 132-136, 159-161, and 170-172, in the `ECE214_2021_Lab3.m` MATLAB® file.)

      i. Plot the measured and simulated magnitude of the output voltage `Vout` in decibels as a function of frequency on one graph.

      ii. Plot the measured and simulated phase of the output voltage `Vout` in degrees as a function of frequency on one graph.

   Compare the simulated and measured frequency response.