

# 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**.

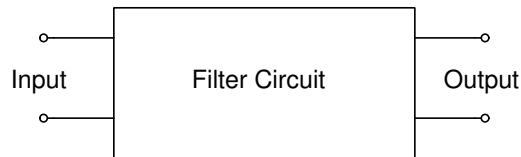


Figure 1: Block diagram of the filter circuit.

## Specifications

1. Input: Square wave with a frequency of 1.25 kHz and a peak-to-peak voltage of 4 V.
2. Magnitude of the input impedance:  $|Z_{IN}| \geq 10 \text{ k}\Omega$ .
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 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](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](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](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.

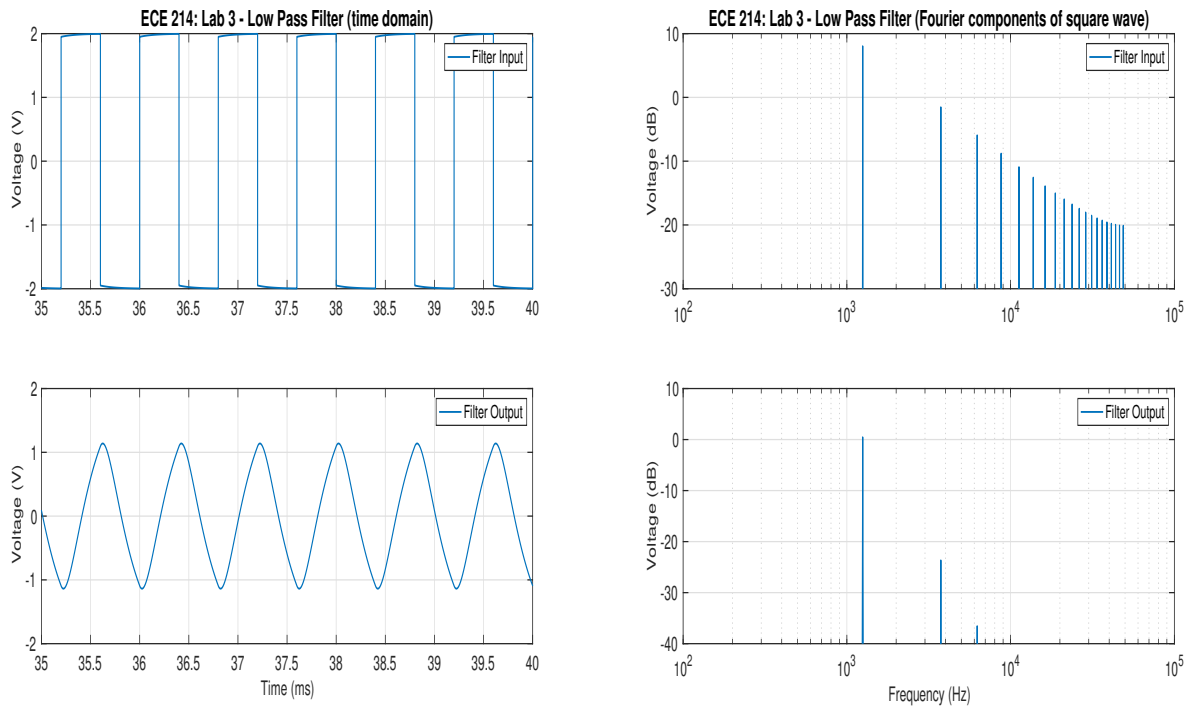


Figure 2: Low pass filter input signal (top) and output signal (bottom) in the time domain (left) and the frequency domain (right).

## 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  $V_{out}$  in decibels as a function of frequency.
  - (b) Plot the measured phase of the output voltage  $V_{out}$  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  $V_{out}$  in decibels as a function of frequency. To convert the output voltage to dB, use:  $20 \cdot \log_{10}(\text{abs}(V_{out}))$ .
  - (e) Plot the phase of the output voltage  $V_{out}$  in degrees as a function of frequency. To convert the output voltage to phase, use:  $\text{angle}(V_{out}) \cdot 180/\pi$ .
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.