Experiment 6: Cascaded RC Low-Pass Filters

Objective

The objective of this experiment is to explore changes to the frequency response of simple passive *RC* low pass filters when loaded with second and third cascaded filter sections.

Background

If two passive *RC* low pass filters are cascaded, the frequency response is not simply the product of the two separate first-order *RC* transfer functions. This is because the ideal single-pole response assumes a zero-output-impedance source is driving the filter and there is no load on the output, i.e., the filter drives an infinite impedance (or open circuit). However, directly connecting the second filter acts as a load for the first effectively changing the combined *RC* time constant. If you try to analyze the cascaded circuit by simply adding phasors you will soon realize the shortcomings of that simple technique. This is where using circuit simulation software is very helpful.

Pre-Lab Exercise

As a pre-lab exercise enter the schematic diagram shown in Figure 1 into LTspice. Three different *RC* low-pass filter sections are included. The inputs of all three filters are driven by the same AC source, V1. Resistor R5 and capacitor C5 form a simple single-pole (1^{st}-order) filter with the output taken at node dB-0. Resistors R3 and R4 and capacitors C1 and C3 form a 2^{nd}-order filter with R4 = R3 and C3 = C1. Two signals from this filter should be plotted – the output of the first section at node dB-1 and the output of the second section at node dB-2. Resistors R2 and R1 and capacitors C4 and C2 form another 2^{nd}-order filter with R1 = 10*R2 and C2 = C4/10. Two similar signals from this filter should also be plotted – the output of the first section at node dB-3 and the output of the second section at node dB-4. This second filter keeps the *RC* time constant the same for both sections of the filter but reduces the loading effect by increasing the value of the second resistor by a factor of 10 and decreasing the value of the second capacitor by a similar factor of 10 (keeping the *RC* product the same). Using a factor of 10 like this is a good rule of thumb to use when designing cascaded passive *RC* filters.

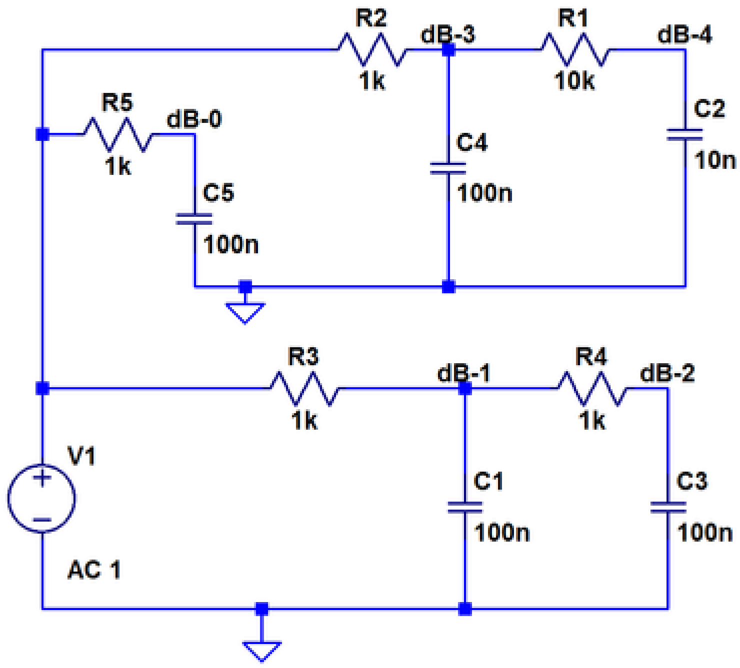

Figure 1. LTspice Schematic Diagram of *RC* Filters.

Run the simulation sweeping the input frequency from 100 Hz to 20 kHz. You should get a frequency response plot that looks something like Figure 2.

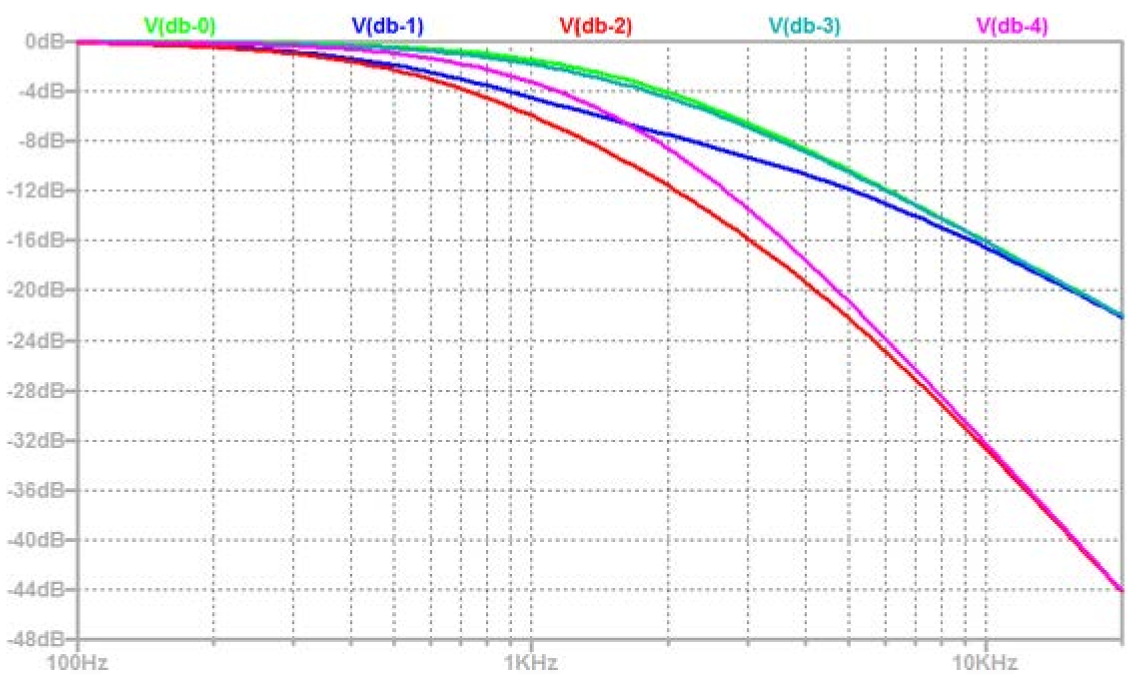

Figure 2. Plot of AC Sweep Simulation Results.

As we can see in Figure 2, the completely unloaded 1st-order filter (dB-0 green line) and the lightly-loaded 1st-order section (dB-3 slightly darker green line) are nearly on top of each other. The loaded 1st-order point (dB-1 blue line) is significantly lower than the other two lines at the frequency of the *RC* time constant. However, all three converge to the same line at high frequencies, 20 kHz or above. The two 2nd-order outputs at dB-2 (red line, loaded) and dB-4 (pink line, lightly loaded) also show significant differences at the *RC* time constant frequency but also converge to the same line at 20 kHz and above. At 20 kHz the response of the 2nd-order filters is 20 dB lower than the 1st-order filters as one would expect.

Equipment Required

- Computer with Windows Operating System
- Fritzing Software (Optional)
- LTspice Software
- Analog Devices ADALM2000 (M2K) Advanced Active Learning Module with Scopy Software and Probe Cable
- Solderless Breadboard, and Jumper Wire Kit
- Components:
    - 3 x 1 kΩ Resistors
    - 10 kΩ Resistor
    - 100 kΩ Resistor
    - 3 x 0.1 µF Capacitors (marked 104)
    - 0.01 µF Capacitor (marked 103)
    - 0.001 µF Capacitor (marked 102)

First-Order *RC* Filter

Hardware Setup

Construct the circuit shown in Figure 3 on your solderless breadboard.

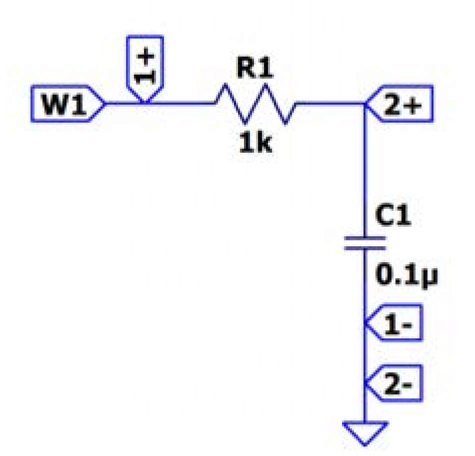

Figure 3. First-Order *RC* Filter.

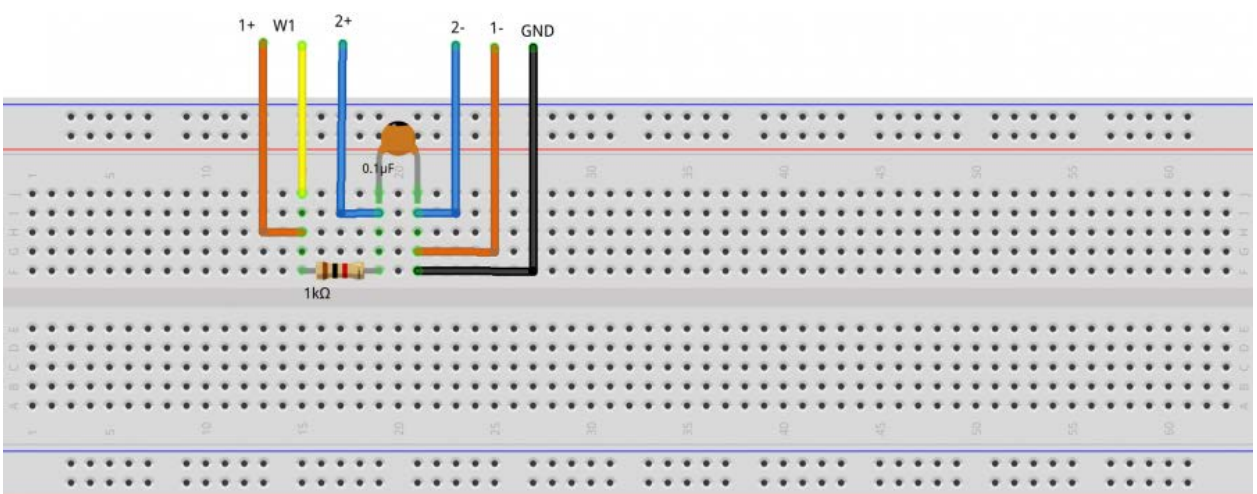

Figure 4. First-Order *RC* Filter Breadboard Layout.

Procedure

With the Network Analyzer, now do a logarithmic sweep. Set the start frequency to 100 Hz and the stop frequency to 20 kHz. The magnitude axis can be set to -50 dB minimum and 10 dB maximum. The phase axis goes from -180 to 90 degrees. Run the Network Analyzer to obtain the frequency response plot of the *RC* filter.

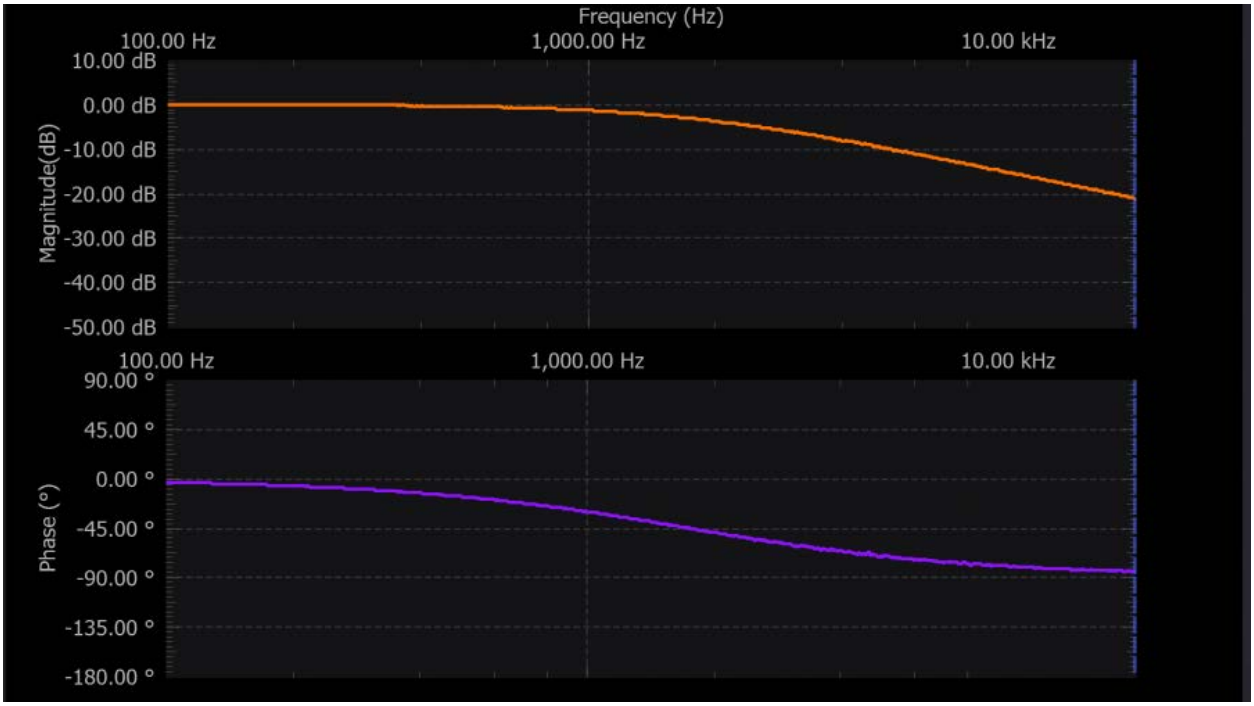

Figure 5. First-Order *RC* Filter Frequency Response.

Second-Order *RC* Filter

Hardware Setup

Add a second *RC* low-pass section to the previous filter as shown in Figure 6.

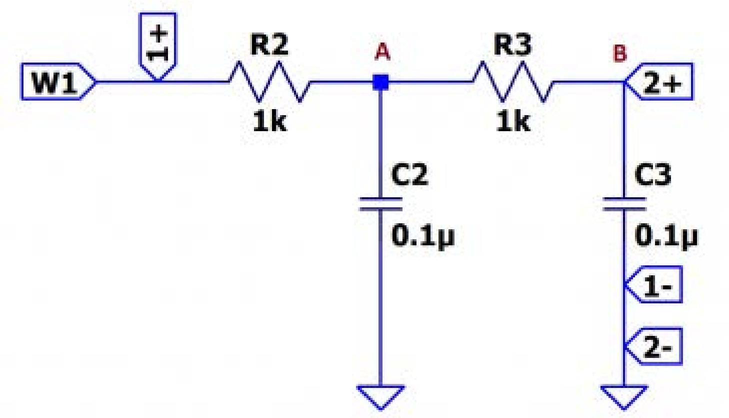

Figure 6. Second-Order *RC* Filter.

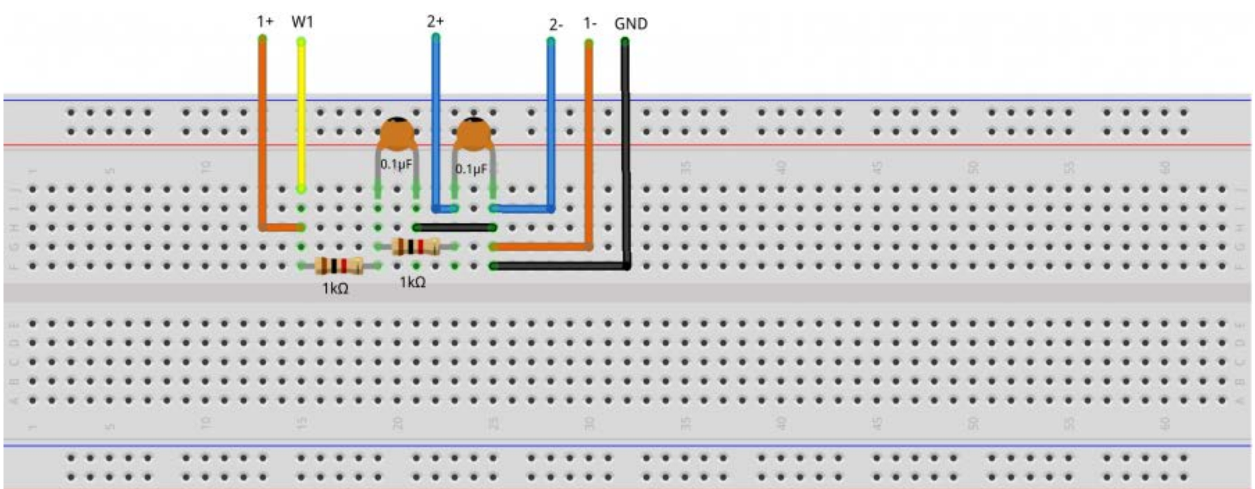

Figure 7. Second-Order *RC* Filter Breadboard Layout.

Procedure

Keep the same settings on the Network Analyzer. Connect Oscilloscope channel 2 after the first *RC* stage (at node A) and do a single sweep. This will be the reference sweep and can be kept on the plot by pressing the Snapshot button in the Reference section of the Network Analyzer General Settings. Move the Oscilloscope channel 2 to node B (after the second *RC* stage) and do another sweep. You should now see the frequency response after both *RC* stages as in Figure 8.

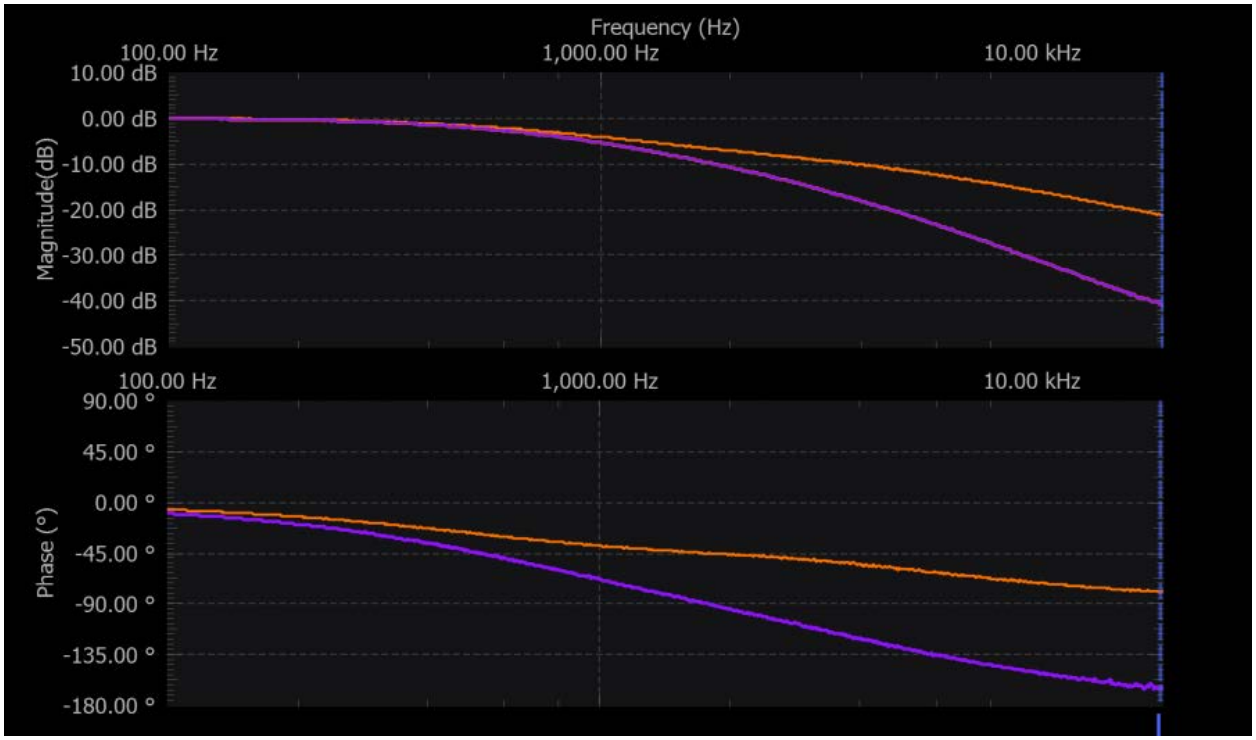

Figure 8. Second-Order *RC* Filter Frequency Response.

Third-Order *RC* Filter

Hardware Setup

As a further extension, add a third *RC* section to make a third-order filter as shown in Figure 9.

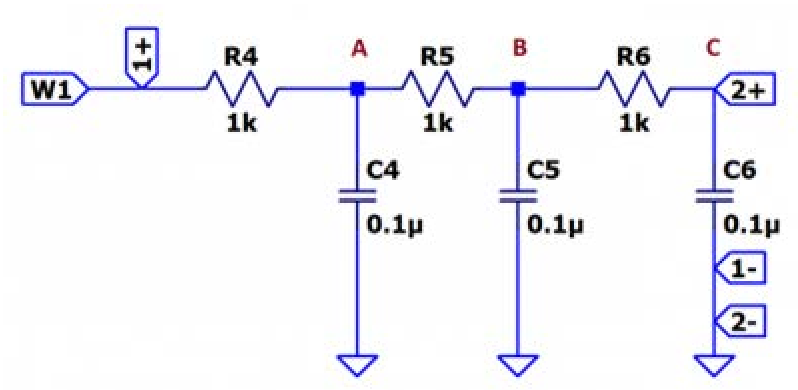

Figure 9. Third-Order *RC* Filter.

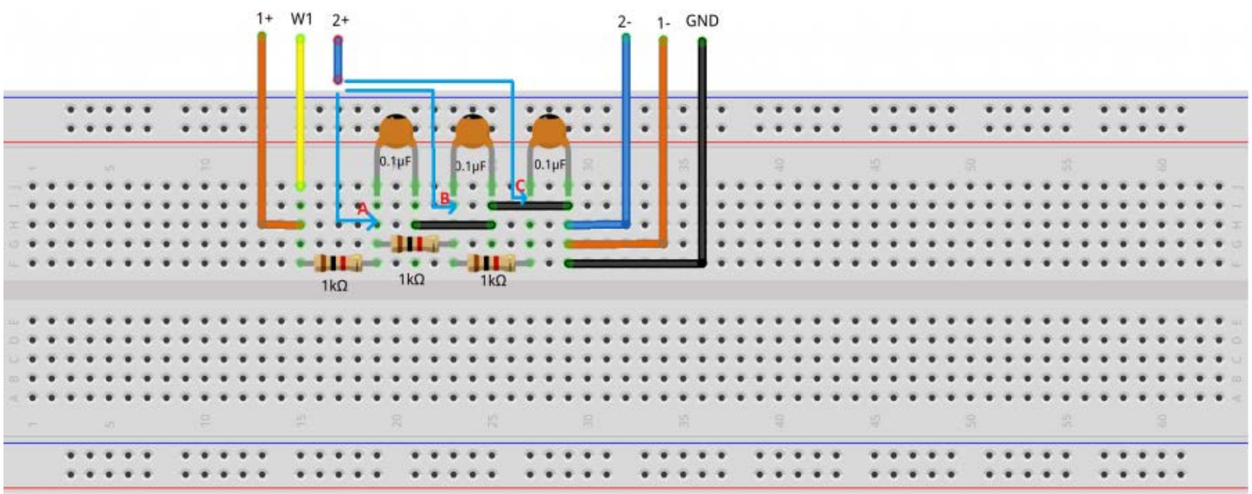

Figure 10. Third-Order *RC* Filter Breadboard Layout.

Procedure

Keep the same settings on the Network Analyzer. Connect Oscilloscope channel 2 after the first *RC* stage (point A shown in Figure 9) and do a single sweep. This will be the reference sweep and can be kept on the plot by pressing the Snapshot button in the Reference section of the Network Analyzer General Settings. Move the Oscilloscope channel 2 to point B (after the second *RC* stage) and do another sweep and press the Snapshot button. You should now see the frequency response after both *RC* stages. To obtain the frequency response of the third stage move Oscilloscope channel 2 to point C and do another sweep. You should now see all three responses as shown in Figure 11.

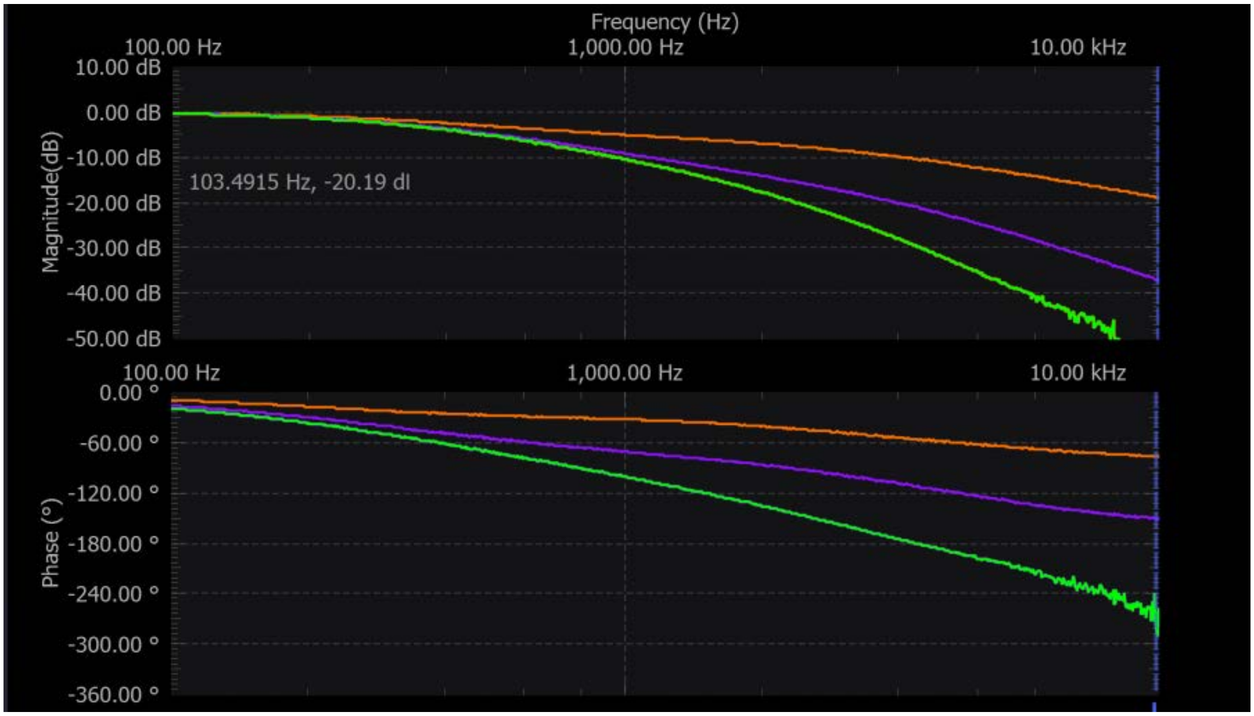

Figure 11. Third-Order *RC* Filter Frequency Response.

Change the components to R4 = 1 kΩ, R5 = 10 kΩ, R6 = 100 kΩ and C4 = 0.1 μF, C5 = 0.01 μF, C6 = 0.001 μF. Repeat the same steps to plot the frequency response for this new cascaded filter.

Questions/Review:

1. What are the differences between the plots for the second-order filter? (One plot was obtained with 2+ at point A and the other with 2+ at point B).
2. What are the differences between the plots for the third-order filter? (The plots were obtained by the same procedure, the third one being obtained with 2+ at point C).
3. How do your measured results compare to your simulation results (as in Figure 2)? Explain any differences.

Report:

Your report should include, at a minimum, screen captures of the Network Analyzer/Oscilloscope displays you obtain, and meaningful discussion and interpretation of your results. It should be assembled in a reasonable report form, not just a collection of pictures. Pretend like you're writing a report for your project manager to explain to her/him what you did and what you think you learned.

Please also include any comments or suggestions you feel may help to improve this experiment for future students.