Analog Elektronik IE1202 - Home Laboratory 2

Student Name:

November 25, 2020

1 Introduction

The following 2 tasks will give you some hands-on experience on basic active filters. You will design, simulate, connect, and measure 1^{st} and 2^{nd} order low-pass active filters.

Very often, the amount of filtering provided by 1^{st} order filters is insufficient since a single pole/zero only results in a slope of ±20 dB/dec. You may need to cascade several 1^{st} order filters in order to reject undesired frequencies. To alleviate this problem, we will introduce the Butterworth filter which can provide a slopes of ±40 dB/dec with a single opamp.

2 Inverting amplifier 1^{st} order Low-Pass filter

2.1 Circuit Theory

For the circuit in Fig. 1 calculate the value of R_2 and C_1 so that the low-frequency gain = 20 dB, and the cut-off frequency f_c = 1 kHz.

Your hand calculation:

Answer R2:
Answer C1:

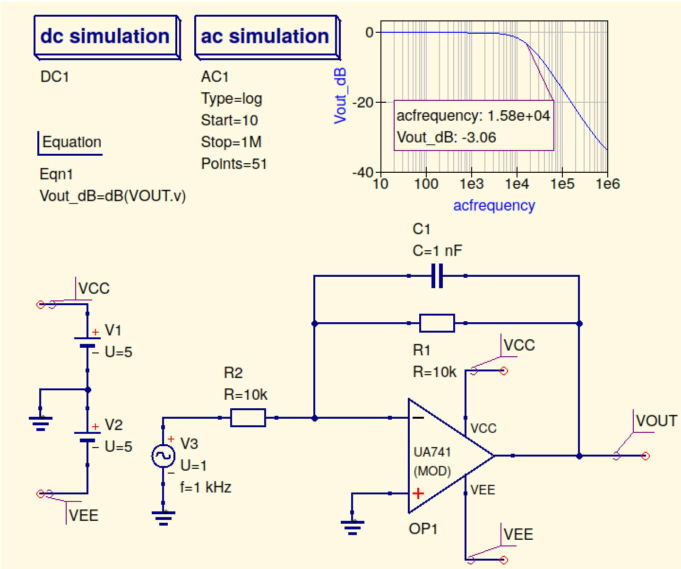

Figure 1: Inverting Amplifier low pass filter

2.2 Circuit Simulation

Open QUCS and set up a schematic like in Fig. 1. Replace R_2 and C_1 with the values that you calculated and run an AC simulation. Save screen-shots of the simulation plots and add them to an appendix at the end of this lab. Is the simulated and calculated transfer function the same?

Comments:

2.3 Circuit Construction and Measurements

Mount the circuit of Fig. 1 on the breadboard. Replace R_2 and C_1 with the values that you simulated in the previous subsection. Connect the arbitrary signal generator and channel B probe to the input. Connect channel A probe to the output. Configure the AWG so that it outputs a sine signal of 100 mV amplitude and frequency of 100 Hz. Enable the AWG and check the amplitude of the waveforms. Change the time base to around 10 ms/div and the number of samples to at least 1 MS. The low frequency gain should be approximately what you calculated before.

Now you will test the frequency response of your circuit and compare it to your simulations. Change the PicoScope to spectrum mode. The first thing to do here is to set the spectrum range. Since the cut-off frequency of your filter is 1 kHz, you may want to set the spectrum range to at least 24 kHz. After that, click on Spectrum Options. Here you may want to change the number of spectrum bins (recommended a large number), and the window function (for instance Hamming, etc. You are welcome to try out all the alternatives!). In addition, in the Y Axis group, change the Y scale to Logarithmic and the logarithmic units to dBV. In the X Axis group, change the X scale to Log 10. Now, got to the top menu, and click on Measurements -¿ Add new Measurement. Select the channel B (input voltage) and for measurement "Amplitude at Peak". Repeat the step for channel A (output voltage). Finally, open the AWG options. Here you can increase/decrease the AWG frequency by using the blue arrows at "Start Frequency". An example of the whole setup is shown in Fig. 2.

Now you are ready to perform a manual frequency sweep. For each frequency that you will test, write down the current frequency and the average values (dBV) of channel A and B. Increase the frequency in steps (click the right arrow to increase the frequency by 1 step) until you cover at least 20 kHz. Remember that the the gain at each frequency can be expressed as Gain_{dB} = $\text{Channel}_{A,dBV}$ - $\text{Channel}_{B,dBV}$. Save screen-shots at 1 kHz and 10 kHz and append them at the end of the lab. Plot the gain vs. frequency on a paper and compare it to what you simulated before. Is the simulated and measured frequency response the same?

Comments:

3 2nd Order Low-Pass Butterworth filter

3.1 Circuit Theory

Fig. 3 shows a Sallen-Key low-pass filter. This filter is very popular due to its simplicity. With only very few components it is possible to produce a transfer function with 2 complex poles in the denominator. Furthermore, if it is desired a frequency response with maximal flatness in the pass-band, the poles can be placed at Butterworth positions by using the following equations:

$$\omega_0 = 2\pi f_0 = \frac{0.707}{R_2 C_1} \tag{1}$$

$$R_1 = 2R_2 \tag{2}$$

$$R_2 = R_3 \tag{3}$$

$$C_2 = 2C_1 \tag{4}$$

where f_0 is the cut-off frequency. The design would typically start by fixing either the capacitors and finding the size of the resistors for a particular cut-off frequency.

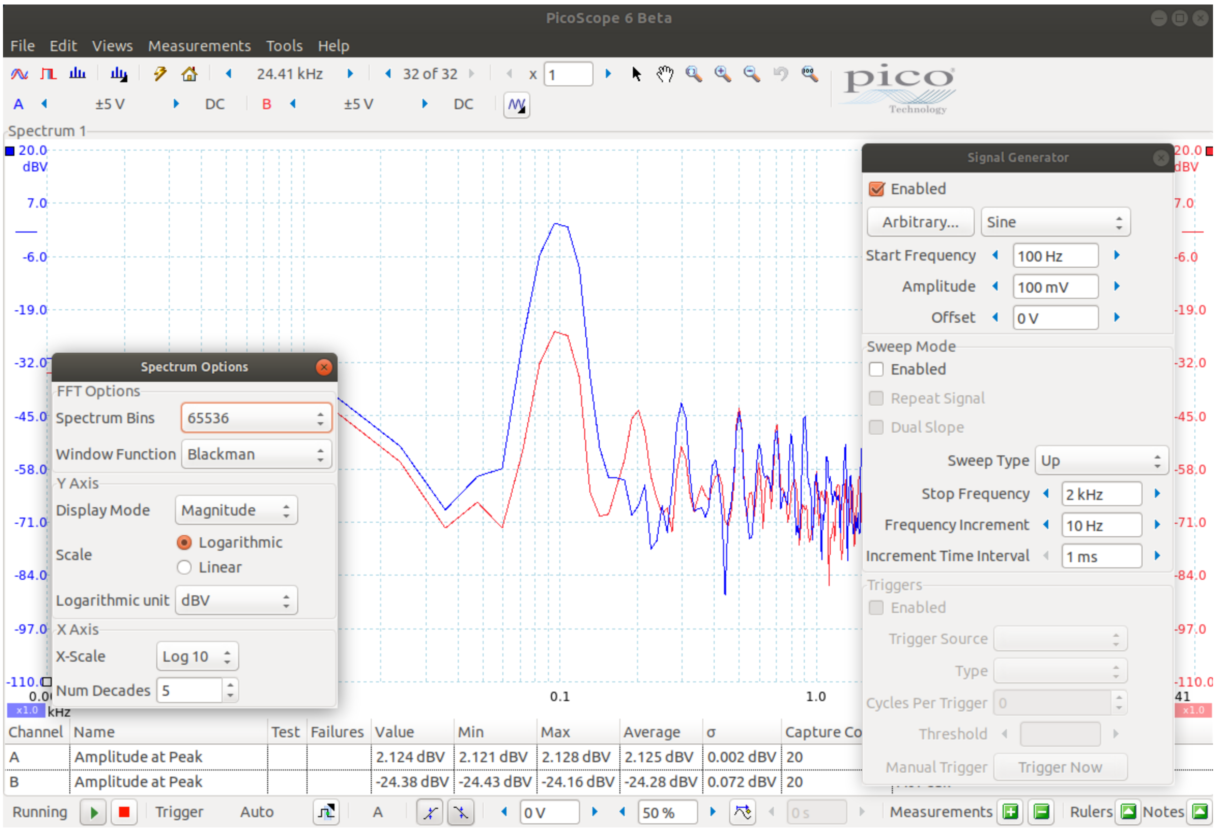

Figure 2: Non-Inverting Amplifier

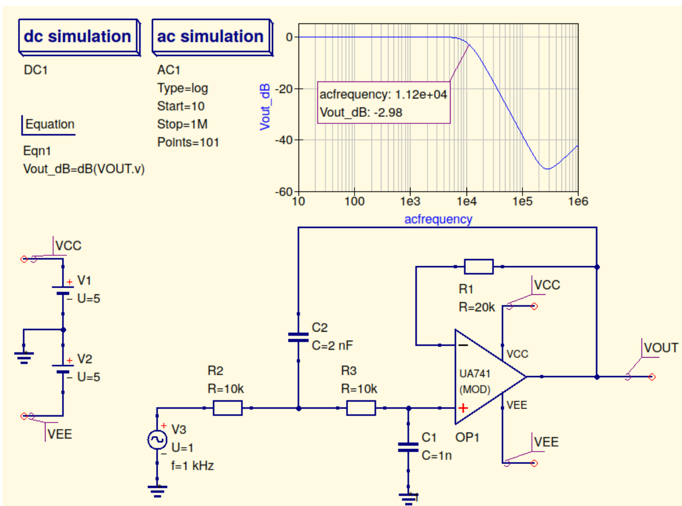

Figure 3: Sallen-Key low-pass filter

Task: dimension the resistors and capacitors in Fig. 3 so that the 2nd order low-pass filter has a cut-off frequency of 1 kHz. Sketch the magnitude of the transfer function (no need to find the mathematical expression, just sketch it).

3.2 Circuit Simulation

Open QUCS and set up a schematic like in Fig. 3. Replace R_1, R_2, C_1, and C_2 with the values that you calculated and run an AC simulation. Save screen-shots of the simulation plots and add them to an appendix at the end of this lab. Is the simulated and sketched transfer function the same?

Comments:

3.3 Circuit Construction and Measurements

Mount the circuit of Fig. 3 on the breadboard. Replace Replace R_1, R_2, C_1, and C_2 with the values that you simulated in the previous subsection. Connect the arbitrary signal generator and channel B probe to the input. Connect channel A probe to the output. Configure the AWG so that it outputs a sine signal of 100 mV amplitude and frequency of 100 Hz. Enable the AWG and check the amplitude of the waveforms. Change the time base to around 20 ms/div and the number of samples to at least 1 MS. Change the PicoScope to spectrum mode and perform a manual frequency sweep as in the previous tasks. Save screen-shots at 100 Hz, 1 kHz, and 10 kHz. Is the measured and simulated gain the same?

Comments:

4 Comments on the laboration

Here you can give as much feedback as possible on this lab. How long time did it take you to complete this lab? Did you find this lab useful to reinforce underlying concepts such as low-pass and high-pass filtering, signal spectrum, etc?