UiT The Arctic University of Norway

Home exam in:	Fys-2008
Date and time:	-Handed out: Tuesday October 3rd (14.00) Submission deadline: Tuesday October 17th (14.00)
Course coordinator:	Frank Melandsø.
Number of pages:	5 including front page
Permitted aids:	All with some restrictions (see point under).
Other:	**NB! Al students must submit individual pdf reports on Wiseflow within the deadline. Similarities in text and figures between reports will not be accepted. It is very important to refer to external source if you include figures and text from these.**

Problem 1

Consider the Wheatstone bridge depicted in Figure 1a) with one sensor arm represented by the resistance R_1.

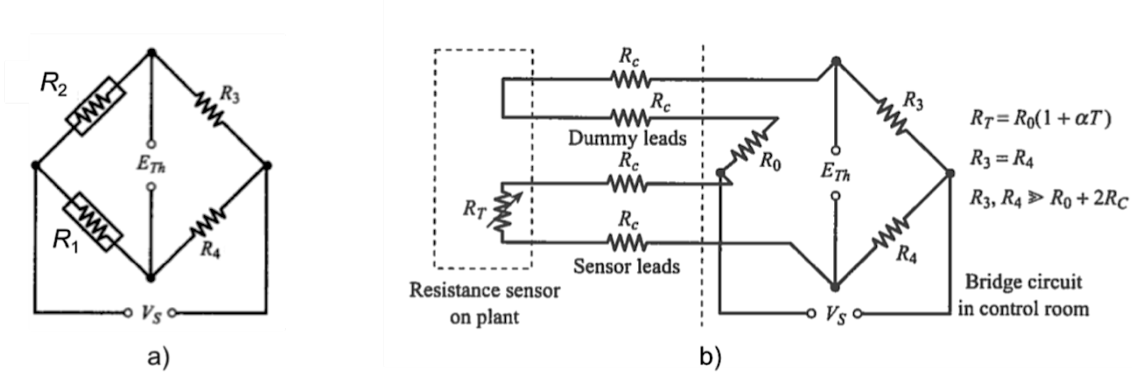

Figure 1 a) Wheatstone bridge with a sensing arm represented by R_1 . b) Four-lead bridge circuit with resistive temperature sensor.

(a) Derive the normalized output voltage E_{Th}/V_s as a function of the four bridge resistances.

(b) Find the value of R1 that gives a balanced bridge.

(c) Consider the bridge circuit depicted in Figure 1b). R_T is a resistance temperature sensor, and R_c is the resistance of the leads connecting the sensor to the bridge circuit. Furthermore, $R_T=R_0(1+\alpha T)$, $R_3=R_4$ and $R_3>>R_0+2R_c$.

Show that $E_{th} \approx V_S(R_0 /R_3)\, \alpha T$.

(d) At which temperature is the bridge in Fig. 1b) balanced? And how can the bridge be made balanced at a different temperature, T_{bal}?

(e) What is the advantage of using the bridge in Fig. 1b) rather than inserting the temperature sensor as R_1 in Fig. 1a) with a randomly chosen value of R_2?

Problem 2:

In this problem we will consider first- and second-order filters made from passive components.

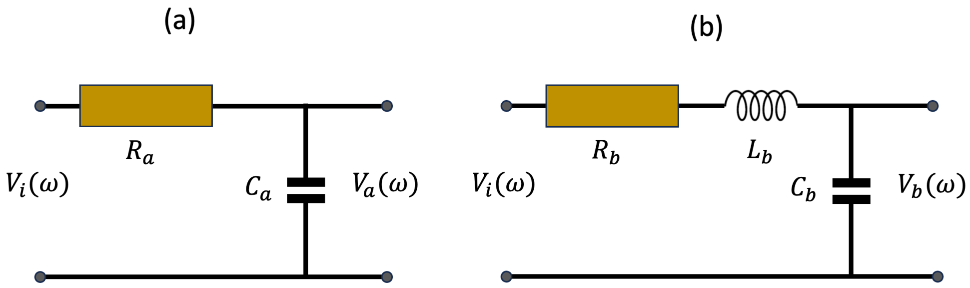

Fig. 2: 1st and 2nd order low-pass filters

(a) We will start by looking at the two circuits show in figure 2 (a) and (b) which both can be used as low-pass filters.
Use the properties of the Fourier transform and the differential equation for the LCR-circuit derived in the lecture, to show that the frequency response for the filter (b) can expressed as the complex function

$$H_b(\omega) = \frac{1}{-\frac{\omega^2}{{\omega_n}^2} + 2i\,\xi\frac{\omega}{\omega_n} + 1}.$$

(b) When using circuit b as a low-pass filter, we will try to design this second order filter so that it's slightly underdamped.

Find a general expression for the amplitude $|H_b(\omega)|$ of $H_b(\omega)$, and show that for $\xi = 1/\sqrt{2}$ the amplitude simplifies to

$$|H_b(\omega)| = \frac{1}{\sqrt{\frac{\omega^4}{{\omega_n}^4} + 1}}.$$

This filter is known in the literature as a 2nd order Butterworth filter.

(c) We will now use LTspice to simulate the filters. Assume that both filters have similar resistors given by $R_a = R_b = 100\ \Omega$. Suggest values for the other components so that both filters get a cut-off frequency at 1 MHz assuming a Butterworth response for filter b.

Do an AC simulation from 10kHz to 100MHz, and view the amplitudes $|H_a(\omega)|$ and $|H_b(\omega)|$ together in the same plot.

Hints for achieving good AC plots: Since we focus on the amplitudes, the phase curves can be turned off. You may also change the default black figure background to a bright color that will be more appropriate for the pdf-report. To verify details in the frequency responses (e.g. cut-off frequencies) it might be useful to turn on grid and adjust ticks/values the y-axis.

(d) Try to change the $R_b = 100\ \Omega$ e.g. by 20% up and down and repeat the simulation. To compare the results, you might consider a parameter study in LTspine to display all amplitude responses together in one plot. Alternatively, you can write the individual solutions to text files, and the use Python (or MATLAB) to read the files and plot the results together.

Discuss the observed results and benefits that you might get from using 2nd order filter instead of a 1st order.

Problem 3:

This problem looks at filters applying an active operational amplifier. These types of filters are normally preferred for handling weak and/or noisy signals coming from sensor elements.

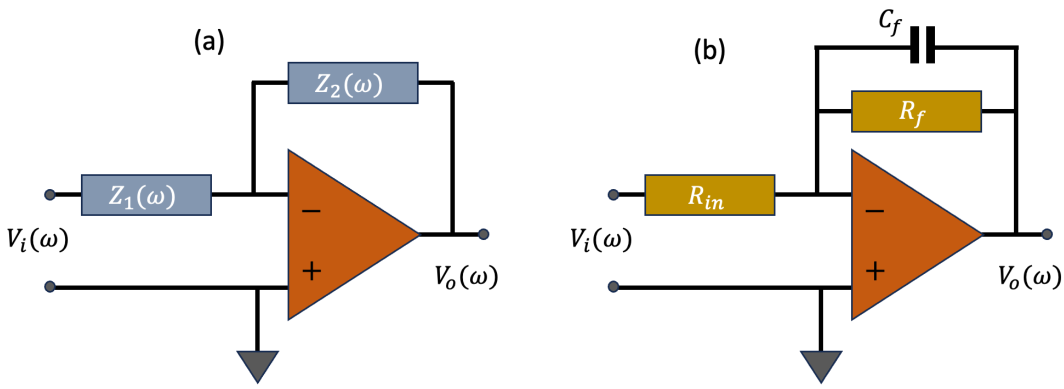

Fig. 3: General filter [Fig (a)] and specific filter [Fig (b)] using an op-amp with negative feedback.

(a) Let us first assume a general inverted amplifier configuration as shown in figure 3a, with two general frequency dependent impedances $Z_1(\omega)$ and $Z_2\,(\omega)$.
Use the needed properties for an ideal amplifier, to show that the amplifier output V_o measured with respect to ground can be approximated by

$$V_0(\omega) = -\frac{Z_2(\omega)}{Z_1(\omega)}\, V_i(\omega)\,. \tag{1}$$

When you derive the above equation, you should comment on made assumptions.

(b) We will then look at the specific filter shown in figure 3b, using a resistor R_{in} for Z_1 and a resistor R_f in parallel with a capacitor C_f for the feedback impedance Z_2.

Find an analytic expression for the frequency response provided by the low-pass filter in figure 3 (b) assuming an ideal op-amp.

(c) You should now simulate circuit b in LTspice, using the Spice model for the high-speed voltage amplifier AD8038 from Analog Devices.

Before starting on the task, you should take a look at the data sheet found here https://www.analog.com/media/en/technical-documentation/data-sheets/AD8038_8039.pdf

Implement the filter 3b in LTspice with the AD8038 spice model. The amplifier model that can be found under “OpAmps” in the component library. In the simulation you should use $R_{in} = 100\ \Omega$ and $R_f = 1\ k\Omega$ giving a low-frequency gain around -10, and adjust C_f so that you get a cut-off frequency at $f_c = 1$ MHz.

Hint: To get bipolar signals from the op-amp, you should connect it to two power input ports to DC potentials +5 and -5 volts. It is also convenient to define LTspice "Net Names" for these power ports as shown in the lower figure here https://electronics.stackexchange.com/questions/529510/how-do-you-use-op-amps-in-a-design-in-ltspice
You will then make the circuit schematics easier to read by avoiding lines between the power points and the DC voltages.
It may also be smart to test the amplifier configuration with only resistors before inserting C_f. The amplifier should then give an almost flat 20dB output up to some upper threshold frequency f_{th} depending on the gain factor, almost as shown in figure 4 in the data sheet. Since f_{th} yields the upper frequency where the amplifier is capable to work, we normally try to design the filter so that the cut-off frequency f_c is considerable smaller then f_{th}.

(d) Repeat the calculations for the filter in (c) using a .tran simulation with a step response as the input. You may here use e.g. 4 us as the simulation time and 0.1 V as the step amplitude.

(e) Suggest a band-pass filter based on the impedance model in figure 3a. Implement you suggestion in LTspice to compute the frequency response and adjust the proposed components to give a band-pass between 10 kHz and 1 MHz.

(f) For the last point we will consider an alternative and fast method for computing the frequency response from a linear filter. The method is based on measuring the time-dependent filter output $v_o(t)$ and computing it's Fourier transform $\hat{V}_o(\omega)$ when applying a short pulse on the input with Fourier transform $\hat{V}_i(\omega)$.

As the system is linear, we can in the Fourier domain assume the input-output relation

$$\hat{V}_o(\omega) = H(\omega)\, \hat{V}_i(\omega)$$

meaning that the frequency response $H(\omega)$ in the Fourier domain can be found from the ratio

$$H(\omega) = \hat{V}_o(\omega)/\hat{V}_i(\omega)\,. \tag{2}$$

If we now assume a delta-function as the input with infinite large bandwidth and Fourier transform $= 1$, we see from Eq. (2) that $H(\omega)$ is given directly from $\hat{V}_o(\omega)$. In LTspice we cannot use a delta-pulse, and must therefore approximate its behaviors using a square pulse with a short on-time τ.

Suggest a theoretical value for τ so that $\hat{V}_i(\omega)$ drops to around -3dB from the maximum value at the frequency 10 MHz. You should then use this τ-value in LTspice's pulse generator with small finite transition time between high and low, to do a .tran simulation of your band-pass filter. Try to adjust the total simulation time so that the frequency band from 1 kHz to 10 MHz can be obtained in the Fourier domain, and compare you're result to the one found in point (e).