Analog Elektronik IE1202 - Home laboratory 4

Student Name:

November 25, 2020

1 Introduction

In this lab, you will build an electrocardiography (ECG) system which records the activity of your heart activity and displays it on a mobile phone or computer. The lab resembles a typical industrial project in which a complex system is partitioned in different building blocks. As analog designer, you will be directly responsible of making sure that the analog front-end block (signal amplification and filtering) accomplishes a set of specifications. The other blocks of the system are already available and will be provided to you.

SAFETY WARNING: The circuit is considered safe only if there is NO instrument connected to the mains electricity (230V) when the electrodes are connected to the body. This means that laboratory power supplies, oscilloscopes, and signal generators MUST NOT be connected to the circuit at the same time that you apply electrodes to your chest. Accordingly, the circuit MUST be only powered by batteries!

NOTE 1: If you do not have an Android device, just omit all the steps related to the App. We will fix that part at the lab during presentation.

1.1 Brief Introduction to ECG

The movement of muscles is caused by electrical excitation of their internal cells. Accordingly, a large amount of useful medical information can be extracted by measuring and interpreting this electrical excitation over time. For instance, each time the heart beats, its muscles produce a waveform similar to the one shown in Fig. 1. Each part of the waveform, which can be identified by a letter, corresponds to a particular event inside the heart. For instance, P corresponds to the atrial depolarization, Q is a downward deflection before a ventricular contraction, R is the peak of the ventricular contraction, S is the downward deflection after the ventricular contraction, and T is the recovery of the ventricles. Cardiologists are carefully trained to detect cardiac problems by observing these waveforms.

1.2 ECG Measurements

Measuring ECG bio-potential signals directly on heart tissues is naturally unpractical. However, it is possible to pick up part of these potentials by measuring voltages directly on the skin at different places near the heart. Fig. 2 shows how this measurement is typically performed. As it can be expected, the heart signals are strongly attenuated. The measured voltages over the skin are very weak, from hundreds of μV up to 1 mV. In addition, most of the ECG's power spectrum is concentrated approximately from 1 Hz up to 100 Hz making it extremely prone to 50 Hz common-mode (CM) noise coming from capacitive coupling to electric appliances and electric power lines surrounding the measurement environment. In fact, common-mode noise at the input of the amplifier can be a very serious issue since it can easily reach levels in excess of 50 mV! Therefore, differential amplification with very good common-mode rejection ratio (CMRR) is necessary. Furthermore, each electrode-skin contact produces a small DC voltage

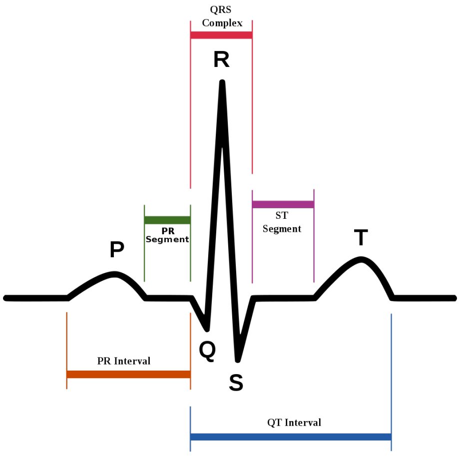

Figure 1: Electrical activity in the heart

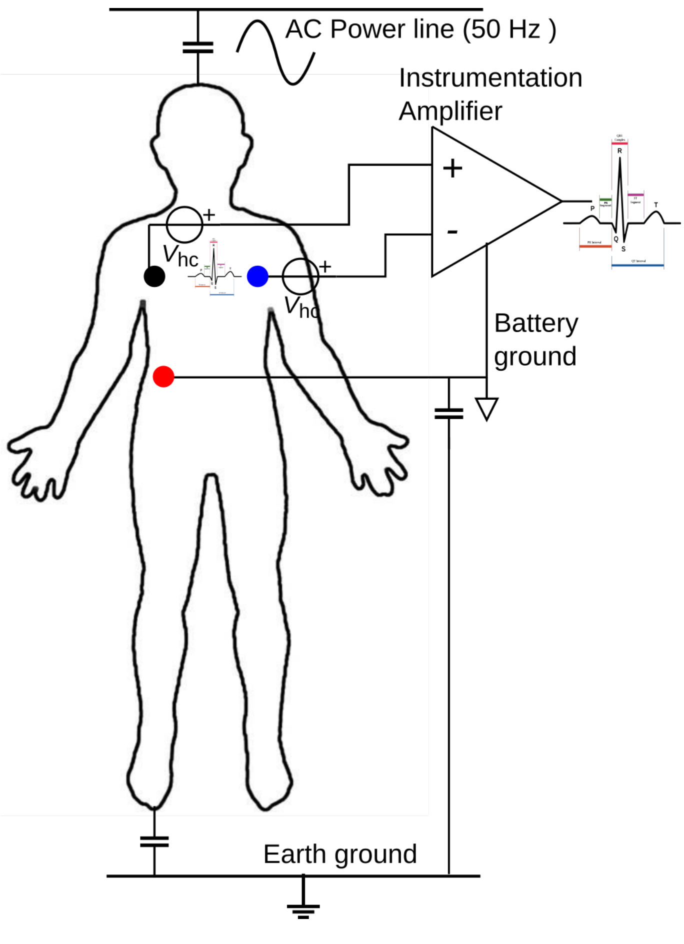

Figure 2: ECG measurement, simplified circuit

which is known as half-cell potential (V_{hc}). In theory, the half-cell potentials are equal and appear as a common-mode voltage which is rejected by the differential amplifier. In practice, mismatches in the electrode-skin contacts produce a DC differential offset which can easily reach several tents of mV.

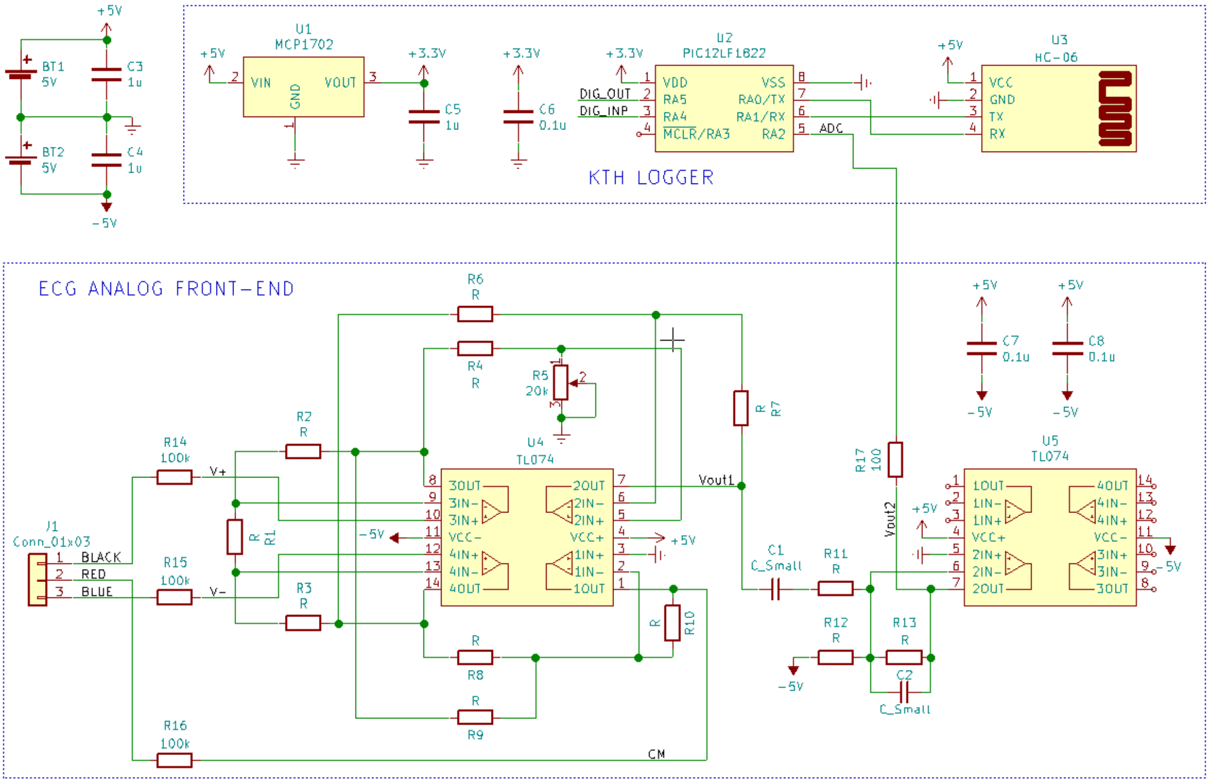

Figure 3: ECG circuit schematics

1.3 ECG Measurement System and Specifications

Fig. 3 shows the complete ECG system which is composed of two main blocks: the KTH logger, and the ECG analog front-end.

The KTH logger is a very small embedded system consisting of a 3.3V voltage regulator, a single PIC microcontroller which has been pre-programmed with the "logger" firmware, and a Bluetooth module HC-06. The PIC has 1 digital input, 1 digital output, and 1 10-bit analog-to-digital converter (ADC) input. The KTH logger communicates to an Android device or a desktop computer running the open-source app "KTH logger" which can be directly installed from the Play Store (you can also download the source code and compile it for your Windows, Mac, Linux). In this lab, you will only use the ADC input in order to record the ECG signal.

The ECG analog front-end comprises all the analog circuits that are necessary to amplify, filter, and condition the signal so that it can be correctly processed by the PIC's ADC . Resistors R_1, R_2, R_3, and Opamps U4.1, U4.2 form the first stage of an instrumentation amplifier (IA). Resistors R_4, R_5 (potentiometer), R_6, R_7 , and Opamp U4.4 form the second stage of the same IA. We will deal with these two stages as a single block and call it "IA". Resistors R_{14}, and R_{15} connect the IA to the electrode ports "BLACK" and "BLUE". They are there for safety reasons! Their high value ensures that currents to/from the body will never reach unhealthy levels if the circuit fails. Resistors R_8, R_9, R_{10} and Opamp U4.3 measure the CM, amplify it, invert it, and inject it back to the body through the resistor R_{16} which is connected to the electrode port "RED". We will call this circuit "CM feedback". This negative feedback loop is an elegant way of performing cancellation of the CM noise. R_{11}, R_{12}, R_{13}, C_1, C_2, and Opamp U5.2 provides additional amplification and band-pass filtering. We will call this circuit "output amplifier/filter". Resistance R_{17} is added to protect the ADC input in case V_{out2} exceeds 3.3V.

The PIC's ADC is able to convert signals from 0 V - 3.3 V (0,V_{DD}) with 10-bit resolution. Since the differential signal is very weak (< 1 mV), it needs to be amplified hundreds of times before the 10-bit ADC can capture it with enough resolution. Furthermore, it is desirable to center the signal at approximately $V_{DD}/2 = 1.65$V so that positive and negative signal variations can be captured. That basically leaves room for a maximum peak voltage of 1.65V before the ADC saturates. In practice, we don't want to work too close to amplitudes that can saturate the ADC since non-ideal artifacts such as electrode location and displacement can result in large amplitude variations. Accordingly, we will take some margins and amplify the signal to around 600 mV so that it can be captured with at least 8 bits of resolution. Besides that, the signal should be high-pass filtered at around 1 Hz just after the IA in order to avoid that the DC offset saturates the output amplifier. In addition, it is necessary to limit the bandwidth to around 100 Hz so that high frequency noise and other out-of-band disturbances do not affect the measurement. Finally, it is necessary to minimize the common-mode noise as much as possible since mismatches due to tolerances in the components will inevitable result in common-mode to differential conversion. Our target will be to keep the output noise below 10 mV_{rms}.

The following table summarizes the specifications of the analog front-end:

Parameter	Specification
$V_{in,diff,max}$	$\pm$1 mV
$V_{out2,diff,max}$	$\pm$600 mV
$V_{out2,DC}$	1.65 V
$V_{noise,out}$	< 10 mV_{rms}
$V_{in,DC,offstet}$	$\pm$50 mV
f_{min}	1 Hz
f_{max}	100 Hz

2 Task

Your task will be to dimension the components in the analog-front-end based on the specifications given in the previous section. In order to help you, a template with the circuit which is ready for simulation is provided for QUCS and QUCS-S (Fig. 4). Note that transient simulation only works properly in QUCS-S! In addition to the analog front-end, the human model from Fig. 2 which includes CM noise coupling is added.

2.1 Circuit Theory

You will start your design by partitioning the gain between the IA and the output amplifier. One of your main concerns here is to make sure that the IA's output should never saturate when amplifying the input DC offset! Since we are using a power supply of $\pm$5 V, you can assume that the amplifiers will saturate at around $\pm$4 V. You need to take some margins, so make sure the DC offset at the output of the TIA will not exceed $\pm$ 3 V. Calculate the gain of the IA and output amplifier.

Your hand calculation:

Gain of IA:

Gain of output amplifier:

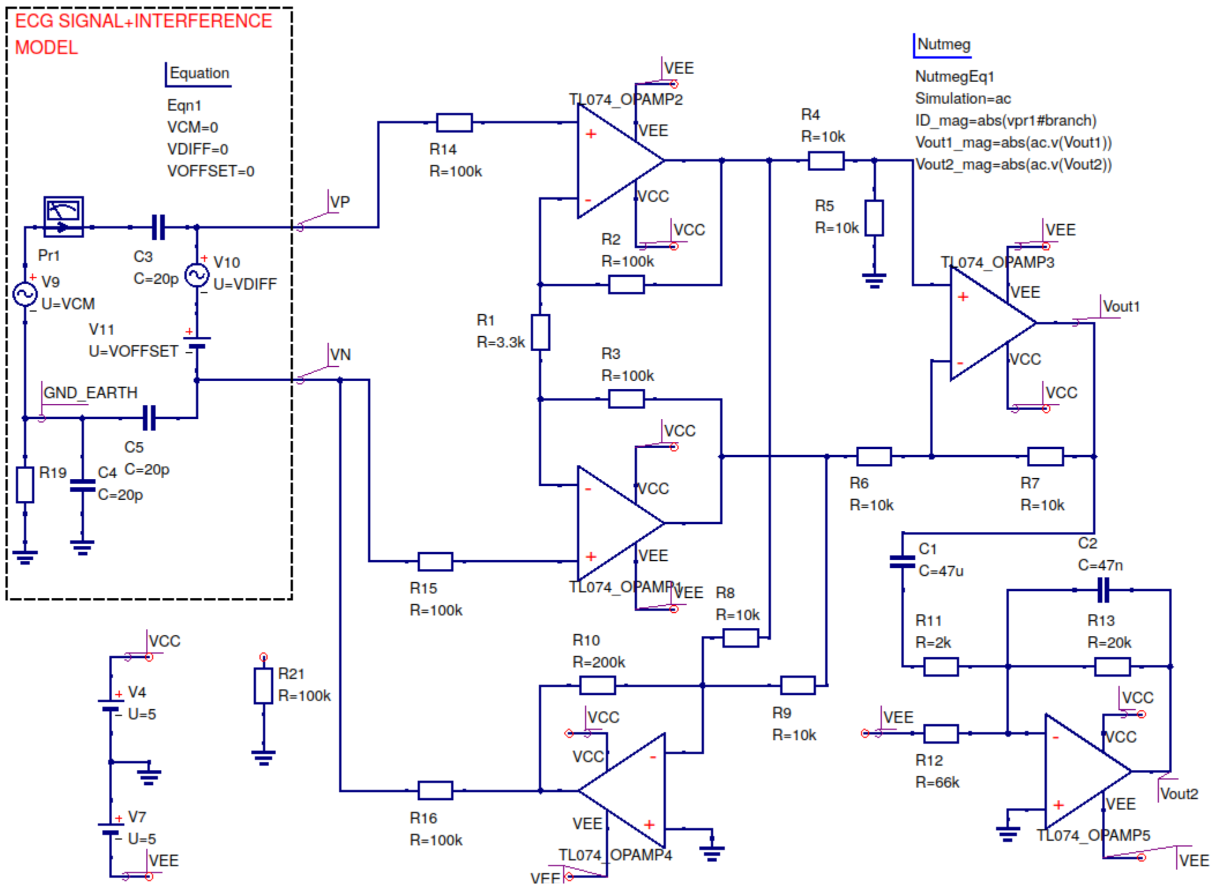

Figure 4: Simulation setup in QUCS-S

Total Gain:

Now, find suitable values for resistors R_1 - R_7. Note that for practical reasons R_5 is a 20 kΩ potentiometer which will be used to calibrate the mismatches in the differential amplifier which are introduced due to tolerances in the components.

Your hand calculation:

The next step is to dimension the components of the output amplifier. Here, you have to consider that this amplifier does 3 things. First, it provides the remaining gain after the IA. Next, it provides band-pass filtering from 1 Hz - 100 Hz. Finally, it raises the signal from 0V to 1.65 V so that the amplified signal is centered in the middle of the ADC range. Find values for resistors R_{11} - R_{13} and capacitors C_1 and C_2.

Your hand calculation:

Finally, find values for the resistors in the CM feedback amplifier. If you look to it closely, you will realize that the circuit is just a summing amplifier. The differential voltage is canceled whereas the CM voltage is amplified and inverted. Remember that this is a negative feedback loop that cancels (or at least mitigates) the CM noise. For stability reasons, it is not desirable to have large loop gain. We will limit the gain of this stage to a factor of 20. Find values for resistors R_8 - R_{10}.

Your hand calculation:

2.2 Circuit Simulation

Open the QUCS-S schematic and change the values of the components with the those that you calculated before. Set VCM, VDIFF, and VOFFSET to 0 and run a DC simulation. Annotate the DC operating point on the schematic. Here you are interested to see if the CM feedback is working (CM voltage will be set to around 0 V), and that V_{out2} is around 1.65 V. If it is not, then you should troubleshoot the problem until you fix it.

Now, you will test that the DC offset does not saturate the output of the IA. Set VOFFSET = 50 mV while keeping VCM and VDIFF = 0 V and run a DC simulation. Annotate the DC operating point on the schematic and check the DC voltages at the output of the first stage of the IA and at V_{out1} (All outputs of the Opamps in the IA). None of these voltages should be close to saturation! Should you see that any of the Opamp's output exceeds ± 3 V, then you will have to reduce the gain in the IA.

The next step is to test the small-signal differential gain. Set VDIFF = 1 V while VCM and VOFFSET = 0 V and run an AC simulation covering from 1 Hz to 1 kHz in logarithmic steps (10 points/dec). Here you are interested to check the gain at points V_{out1} and V_{out2}. Are the simulated gains the same as the ones you calculated? Are the cut-off frequencies of the passband filter as you calculated? If the answer is no, then you should find the reason and correct it.

After you have tested the small signal gain, you can move to the last step which is to test the circuits with a large signal simulation (transient simulations). First, you will test if your CM feedback circuit can handle 230V AC power lines nearby your measurement! Set VCM = 230 Vrms $\times\sqrt{2}$ = 325 V and leave VDIFF and VOFFSET = 0. Run a 200 ms transient simulation and plot the voltage waveforms at the nodes VP and VN. If your circuit is working properly, then you should see only a few mV, CM noise (50 Hz) at these nodes. This CM noise will be suppressed by the differential amplifier at the IA's output (provided that R_4 - R_7 match well).

In order to understand the importance of the CM feedback in our design, we will disconnect R_{16} from VN and run the same transient again. This is equivalent to remove the red electrode in Fig. 2. Now the ground reference of our circuit is floating with respect to the body and you should see a CM signal of approximately 150 V amplitude at the inputs! Clearly, we need the red electrode in place! But why not just connect it to the ground of our circuit? Let's try it. Connect R_{21} to VN and run the simulation again. Now the waveforms at VP and VN should show around hundred mV CM noise coupled from the power line. Although not necessarily a catastrophic situation, the tolerances in the components will introduce CM to differential conversion which will be much harder to mitigate. We will disconnect R_{21}, and reconnect R_{16}.

Now we will add our differential signal. Set VDIFF = 1 mV, VCM = 325 V, and VOFFSET = 0. The differential signal is configured as a sinusoidal source at 10 Hz. Run the transient simulation and observe the waveforms at nodes VP, VN, V_{out1}, and V_{out2}. You should see at

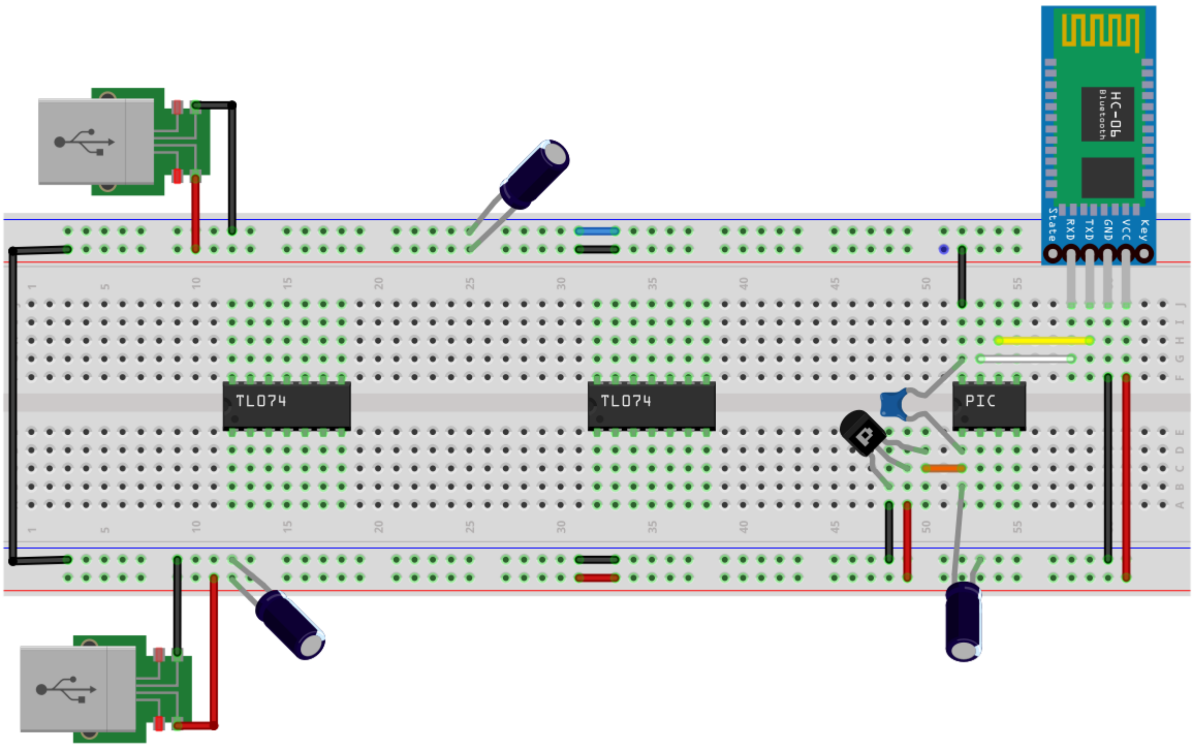

Figure 5: KTH Logger

VP and VN the 1 mV differential signal accompanied by a somewhat larger CM noise at 50 Hz. At V_{out1} you should see only the 10 Hz signal amplified and center at 0V. At V_{out2} you should see the 10 Hz signal further amplified and center at around 1.65 V.

Finally, add the DC input offset. Set VDIFF = 1 mV, VCM = 325 V, and VOFFSET = 50 mV and run the transient simulation again. V_{out1} should have changed the DC output level. V_{out2} should be the same as before. Take a screen-shot of your transient simulation and append them to the end of this lab. Take also a screen shot of your final schematic showing all the components in Fig. 4.

2.3 Circuit Construction and Electrical Characterization

Mount the circuits of Fig. 3 in a breadboard. Since there is limited space in the breadboard, we recommend to connect the devices as shown in the next figures. Replace the resistors' values with your own solution! Don't forget to use colored cables: red = +5V, blue = -5V, and black = 0 V/ground (do not confuse with electrode nets "BLACK", "RED", and "BLUE" that should NEVER be connected to the power supply nets!). First, you will only mount the KTH logger and test it with the android App (download it from the Play Store, or compile it from source for your platform). Fig. 5 shows the KTH logger.

Power the circuit with the 5V batteries. Enable Bluetooth in your Android device or computer and pair it with the HC-06 device (pairing code: 1234). Then open the KTH logger App, and click on the "Connect" tab. After that, select "Bluetooth", and press the button "Discover BT". Click on the drop-down menu and select the HC-06 device from the list. Click connect and wait until the red led in the HC-06 modlule stops flashing. The status message should change to "Connected". Then go to the "AT" tab. This is a small terminal where you can send commands to the KTH logger. Send the character "t". If everything is correctly connected, then you should receive the answer "Ok". Close the App and disconnect the batteries.

Now you can start mounting the analog front-end. You will mount/test one stage at a time. First, start with the IA as shown in Fig. 6. Before connecting the potentiometer, set its value to

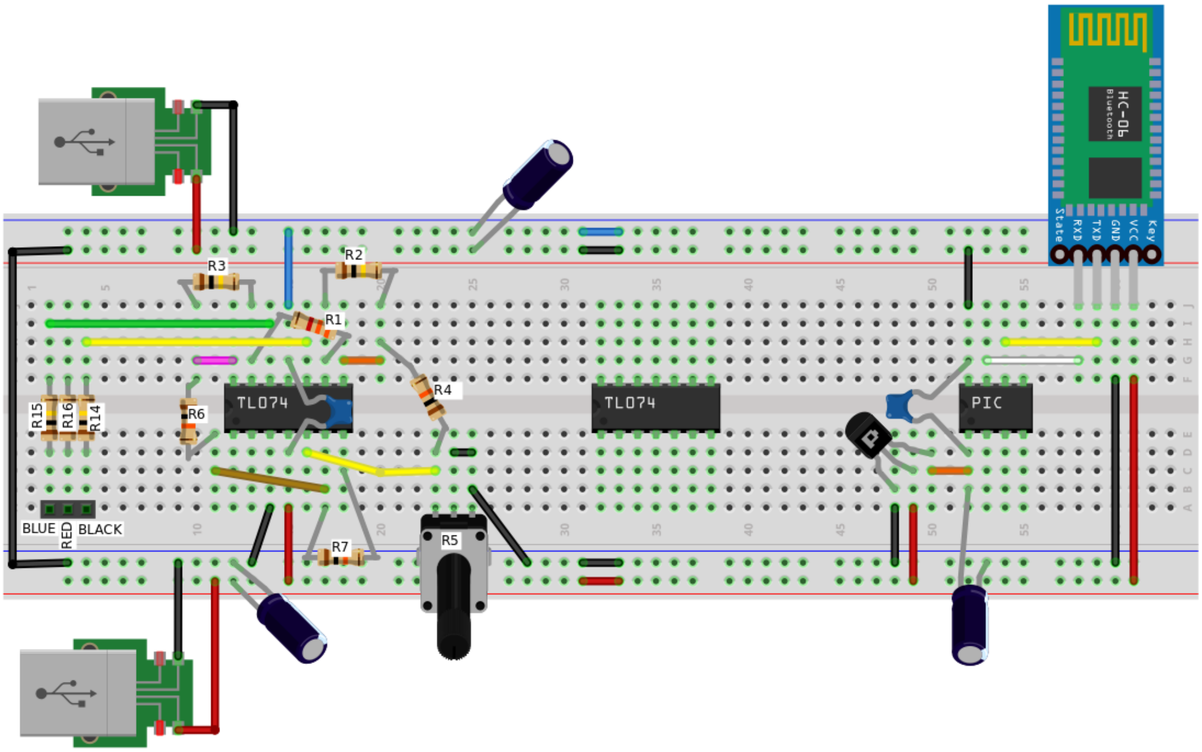

Figure 6: KTH Logger + IA

approximately what you simulated. You will use the PicoScope to test the functionality of the circuit. You basically need to test the same things that you simulated in the previous section and trouble-shoot your circuit if something is not working.

You will start by testing the DC operating point of the circuit. Connect the electrode port “BLACK” and “BLUE” to ground, and then connect the batteries. The DC offset coming from the Opamps at the first stage will be multiplied by the gain and appear at V_{out1}. You can probably tolerate here a few hundred mV. Should the output offset exceeds 1 V, then you may need to reduce the gain of the first stage. Remove the batteries and disconnect the electrode ports.

Next, you will test the differential gain for DC and AC signals. Connect the PicoScope AWG’s signal output to the “BLACK” port and the PicosScope AWG’s gound to the ground of your circuit. Also, connect the “BLUE” port to ground. Connect the PicosCope channels A and B to the “BLACK” port and V_{out1} respectively. You will first test the effect of DC offsets coming from the electrodes. Set the AWG with a sinusoidal signal generator of 0V amplitude and 0V DC offset. Enable the AWG and check the waveforms. At this point you should see exactly the same output offset as before. Now we will add DC offsets coming from the electrodes. Increase the DC offset of the AWG from 0 V to 50 mV in steps of 10 mV and check V_{out1}. How much electrode offset can your circuit tolerate before the output goes above ±3 V? Should the output exceeds considerable 3V, then you may need to reduce the gain of the IA. Now, you will test the AC gain. Change the AWR settings so that the DC offset is 0V, the amplitude of the sine signal is 20 mV, and frequency 30 Hz. Check the waveform amplitudes and calculate the gain. Is it the same as you simulated? Remove the batteries and disconnect the electrode ports.

Now, you will mount CMFB components and thest the CM features of your circuit. Add the connections and components as shown in Fig. 7 (replace the components with the values you simulated!). In this part, you will first test and minimize the common-mode gain of the IA. As mentioned before, the common-mode noise is a serious issue and you want to minimize it as much as possible. Connect the “BLACK” and “BLUE” ports to the PicoScope AWG’s

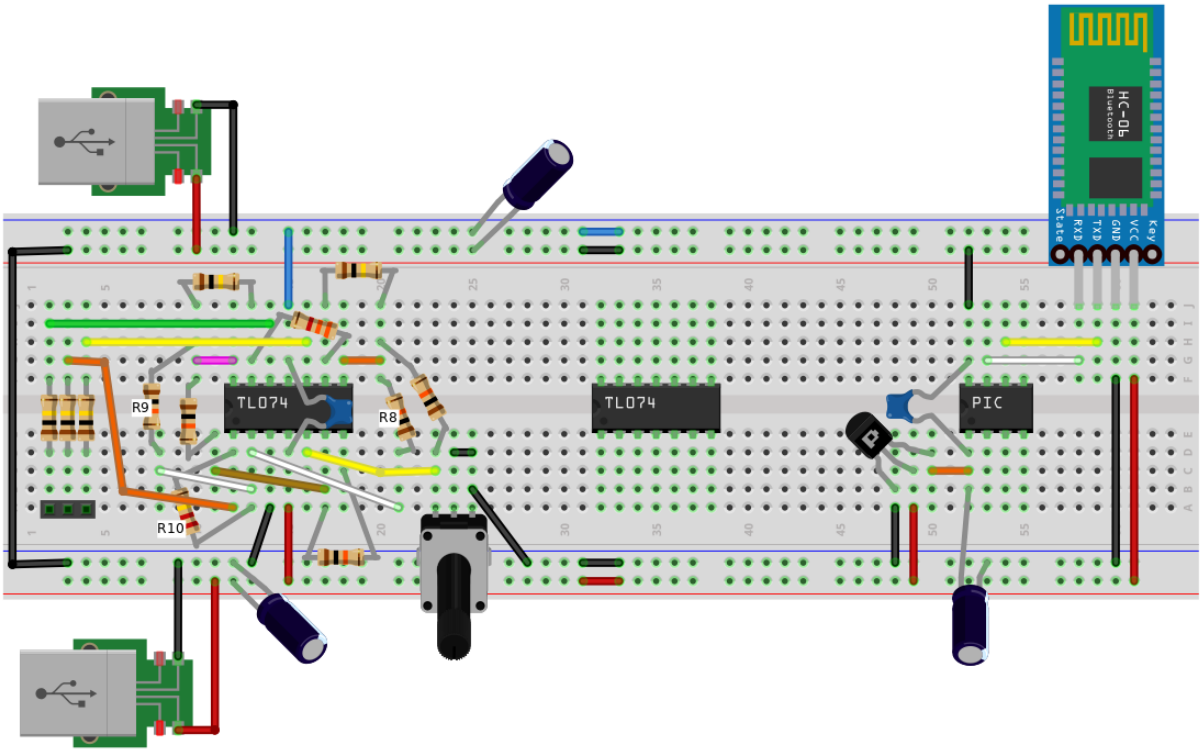

Figure 7: KTH Logger + IA + CMFB

signal output. Connect the PicosCope AWG's ground to the ground of your circuit. We will first check the CMFB circuit. Set the AWG with a sinusoidal signal generator of 0 V amplitude and 20 mV DC offset. Connect the PicoScope channel A to the AWG's signal output, and channel B to the electrode "RED" which carries the CMFB output. Check the magnitude and the sign of the signal. Does the measured and simualted CMFB is the same? Finally, you will minimize the common-mode to differential conversion by tuning the potentiometer R_5. Set the AWG sinusoidal signal generator to 500 mV amplitude and 0 V DC offset, and frequency 50 Hz. Move the PicosCope channel B to V_{out1} and change its settings to AC coupling (you may want to change its resolution to 12 bits also!). You most probably will see a rather large 50 Hz signal at V_{out1}! Now start moving the potentiometer until this signal is minimized. What is the AC rms level of this signal after you minimize it? For completeness, you should check again your differential AC gain just to make sure it has not changed. Remove the batteries and disconnect the electrode ports.

the next step is to add and test the output/filtering stage. Mount the components as shown in Fig. 8 (replace components' values with the ones you simulated). We strongly recommend you to disconnect V_{out1} and V_{out2} from the other blocks so that you can test this circuit individually. First check if the DC operating point is correct. V_{out2} should be centered at around 1.65 V. Then use the AWG to check the gain and cut-off frequencies of the band-pass filter. Compare your measurement with the simulation. Should you see large variations, then you should troubleshot the problem before continuing. Remove the batteries.

Finally, you can reconnect V_{out1} and V_{out2} to the other blocks. Testing the circuit as it is now can be very challenging since the analog front-end has a lot of gain, and just a few mV AC at the differential input electrodes will saturate V_{out2}! Connect the Picoscope AWG for differential AC tests, and set the AWG amplitude to 5 mV, frequency to 60 Hz, and DC offset to 0 V. Change the PicosCope channel A to the AWG's output, and channel B to V_{out2}, and measure the AC gain. If the output saturates, try to reduce the amplitude to 3 mV. Your measured gain should be similar to what you simulated!

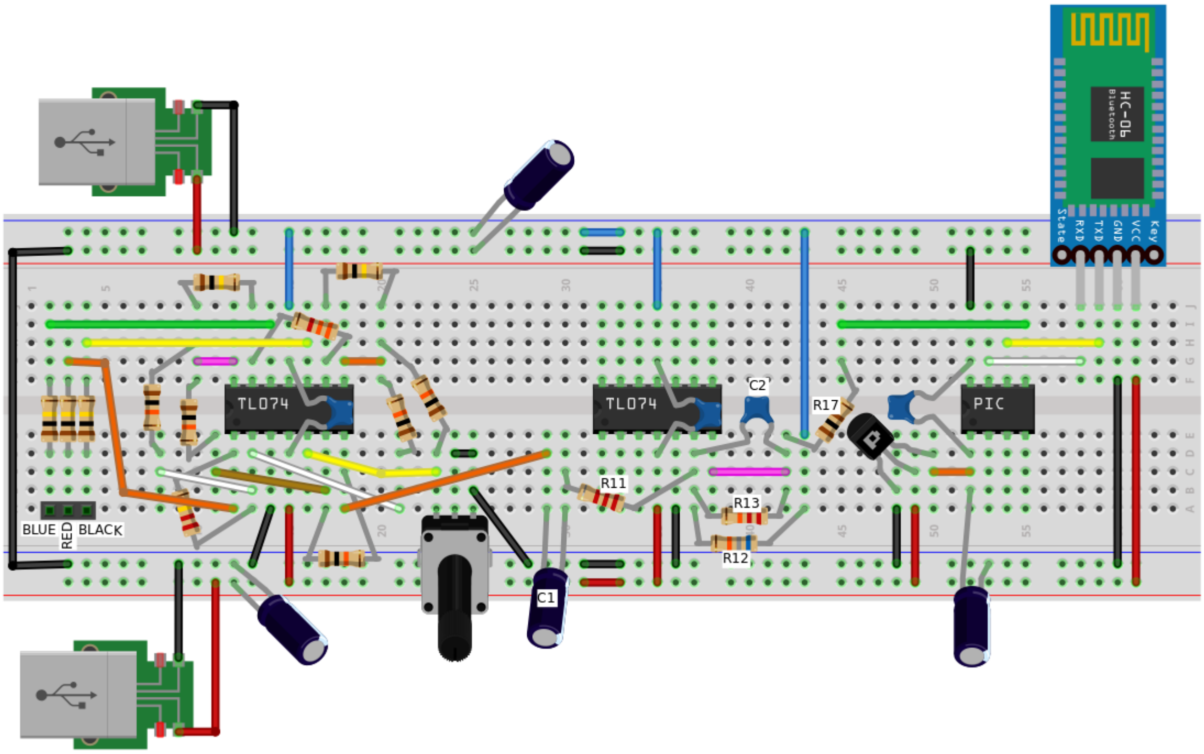

Figure 8: KTH Logger + IA + CMFB + Output amplifier/filter

2.4 ECG Measurement

WARNING: since this is the first time this lab is run, this step will be done with supervision of an assistant!

Disconnect the PicoScope, batteries, and any other external instrument that may be attached to the circuit. No part of this circuit should be connected in any form to any instrument which is powered from the AC power mains! The circuit is completely safe only when it is powered by batteries. To measure your ECG, follow the next steps:

1. Connect the ECG electrodes to the black-red-blue cables provided in the in the kit

2. The electrodes will be placed in the locations shown in Fig. 2. Make sure you identify the position and color of each electrode! Before connecting the electrodes, clean the skin with the "wound cleanser" (cotton with saline solution) and dry it with a napkin or cotton. The electrodes are of the "wet" type and come with conductive gel in the middle. To apply them, just remove the plastic adhesive that protect the contact surface and place them on the skin with a soft massage making sure there are firmly attached to the skin (don't drop the plastic adhesive since you can still reuse the electrodes a couple of times!).

3. Connect the cable to the circuit

4. Connect the batteries to the circuit

5. Start the KTH logger App and establish connection with the HC-06 BT module.

6. Go to the "AT" tab and send the character "s" to start the oscilloscope function. You will see that data start to be collected from the KTH logger. Now, go to the "Plot" tab. You will now see the ECG being displayed in real-time! You can put your device in landscape mode and enjoy! You can try to Smooth the plot or use antialising filtering if you want

("Config" tab). Take a screen shot of your ECG and append it to the end of this lab. To stop sending data, go to the "AT" tab and send "S".

7. Disconnect from the BT and remove the batteries.

3 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 common-mode, differential mode, offset, pass-band, etc?