Design, development and testing of a analog ECG amplifier

Bjørn-Jostein Singstad, *Student, UIO,*

***Abstract*—Electrocardiography (ECG) is used to measure the electrical activity of the heart. This paper describes the development of a full analog ECG frontend with a notch filter. The design of the printed circuit board (PCB) is done by using Cadstar. The components where attached to the board manually with the stencil printed solder paste. Finally the product where tested against another ECG-device (AD8232)**

***Index Terms*—ECG - Electrocardiograph**
PCB - Printed Circuit Board
UIO - University of Oslo
SMD - Surface Mount Device
RL - Right Leg
RA - Right Arm
LA - Left Arm
V - Voltage
Hz - Hertz

I. Introduction

ECG is the golden standard when it comes to measure electrical signals from the heart. A motivation for creating this device is to make it better then the lower cost ECG devices on the marked. Before starting the design process, some key requirements where made (table 1). These requirement are not just to ensure good performance of the product, but also to provide an easy test setup after assembly and to ensure that the product can be packed in a satisfactory manner.

The schematics of the ECG was given from the supervisors of UIO. This circuit has been realized as part of a PCB earlier and is also verified that it works. This project had to be reduced in complexity so it had to be modified and thus there was not just to copy the original schematics.

The biggest challenge in this project was told to be the notch filter. The notch filter is used to eliminate or reduce the effect of the 50Hz power-line interference[2]. In this design, the notch filter uses two capacitors, C12 and C13, that are set as close to the amount of Farad as possible (figure 8 in appendix)

II. Methodology

A. Target requirement specification

The first project milestone of this project was to think carefully through how to use the device and write down some key points. Table 1 shows the target requirement that was drafted before the design process started.

B. Cadstar

Cadstar is a PCB design software from Zuken where the complete design process could be done in the following four steps; schematic design, PCB component placement, PCB routing and create production files.

TABLE I
Target requirement specification for the ECG device

Functionality	Target specification	How to verify that the target is reached
On/off-button	Power on/off	Verify that there is no voltage in the circuit when the button is in the off position
LED	Light on/off	Turns on/off when on/off button is pressed
Power supply / battery	9V battery	The circuit works when the battery is connected and in the on position
Input signal	3 small holes. Wires can be soldered to station where they can be connected to the electrodes	Measure with a multi-meter if contact between electrodes and the input on the PCB is established
Output signal	2 holes (signal and ground). Attach pins so it will be easy to access with a oscilloscope	Measure with oscilloscope and evaluate the output value
Test-point between instrumental amplifier and notch filter	In order to do debugging on the PCB it should be a test-point between the instrumental amplifier and notch filter	Measure the test-point output signal with an oscilloscope.
Measuring signals	Get input signals from a human and a signal generator	Evaluate if the instrument is able to detect electrical heart activity by using a heart signal simulator.
Filter noise from mains (50Hz)	Notch filter	Evaluate the output signal with an oscilloscope
Holes in corners	One hole (4mm) in each corner to be able to attach the PCB to a box.	Ensure that the screws fit
Components and placement	The components should be selected and positioned to capture as little noise as possible. The measured signals should be within 150mV. So components must be selected from this.	Evaluate the output signal with an oscilloscope

1) Schematics design: The schematics was given by the UIO supervisors, but had to be modified to keep the project within the limits of the relativly small component library at UIO. Components were added to the schematics in parallel with component placement and routing. Cadstar allows the user to do this quite efficiently.

2) PCB component placement: From the start of the component placement, I chose to group the components the way that the signals went from left to right. I also tried to place the components, in the first part of the ECG amplifier, close together to reduce the impact of noise.

The PCB is a four layer PCB and all components are put on the topside of the PCB. The two middle layers are used for ground (GND) and 5 volt. The bottom layer is mainly used to route 2.5V, but also some of the signals from the top side layer.

3) PCB routing: PCB routing was the most time consuming during the design process. At this stage of the design process it was necessary to go back and forth between placement and routing to fit everything on the board. 45 degrees routing where used where it was appropriate.This occupies less space and make the design look tidier [3]

4) Create production files: The PCB files was sent to the supervisors who created production files that was sent to a professional PCB manufacturer.

C. Assembly

To attach component to the PCB, a solder paste was added to the board. This was applied by using stencil print [1]. The components could then be added carefully to the board by using tweezers and an optical microscope. The board was then put into an oven for about 15 minutes followed by a visual inspection in the optical microscope. Some components had to be flipped or moved, which was done using a solder tweezer. Some of the big or complex components had to be soldered manually, like C12, C13, CN1 and CN4 (appendix).

D. Test

1) Initial testing: Several testing points where added at strategic locations to test the circuit functionality. I chose to use pins that where easy to access with probes from multimeter and oscilloscope. The functionality test of the circuit was done directly after the assembly, assisted by the supervisors at UIO. A heart simulator was attached to the inputs, LA and RA, and a 9V battery was connected to the power input. A digital oscilloscope was used to read the output from the circuit. Later the circuit was connected to a human body by using electrodes on both side of the chest. To read the output, a battery driven oscilloscope (9V) where used instead of the one connected to mains. This is was done to avoid big fault currents, from mains going trought the chest.

2) Further testing: To be able to tell whether this ECG was as good as the commercial devices it had to be tested and compared to a commercial ECG (AD8232 ECG-chip). The devices were equally tested in the electronic lab which was assumed to be noisy in terms of the alternating magnetic field which is generated from mains. This electric noise in the lab was used to test 50Hz noise rejection.

The ECG, designed in this project, has only 2 leads. This means that it has no electrode to pick up the reference voltage from the rigth leg (RL), normally found on commercial ECG devices. Finding the reference point is solved by taking the average of the two inputs.

AD8232 has 3 leads where the third lead is allocated to pick up the reference voltage.

All measurement were carried out in the first lead direction/vector according to Einthoven's triangle when the two devices where tested against each other [4]. A data logger was made by an Arduino with a SD-shield. The sample rate was more than 50 samples per second.

III. Results

Before the PCB design files were sent to production, the device could be visualized by generating a 3D model from the design files. This was helpful in determining location on some of the major components, such as battery input and signal output. Figure 1 shows the finished PCB after the assembly. Figure 2 shows the 3D model that was created in Cadstar from the design files.

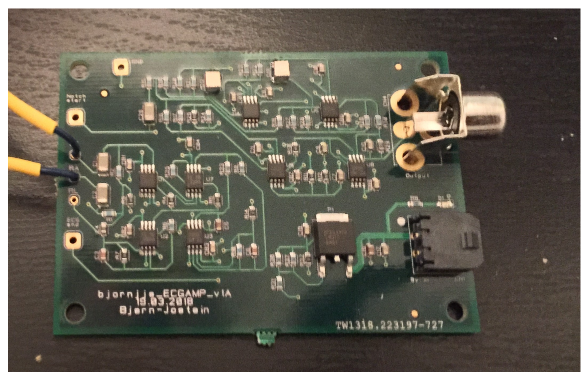

Fig. 1. The finished product

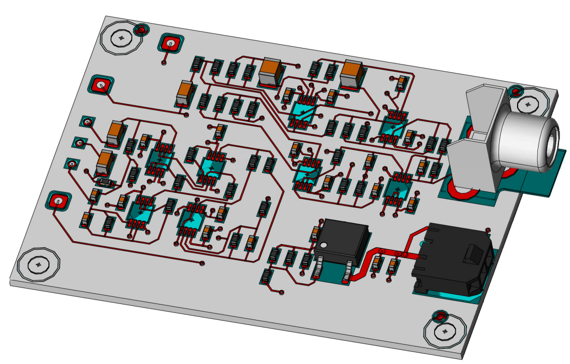

Fig. 2. 3D model of circuit board

For the initial tests, Table 1 was used as a checklist. The device fulfilled all requirements except the on/off-button, which was not included in this design.

In the test agains AD8232 the to first measurement was done with all available electrodes connected to the body (Figure 3 and 4). One last measurement was done without the RL-lead connected to the body (Figure 5).

Fig. 3. AD8232 with 3 leads connected to the body

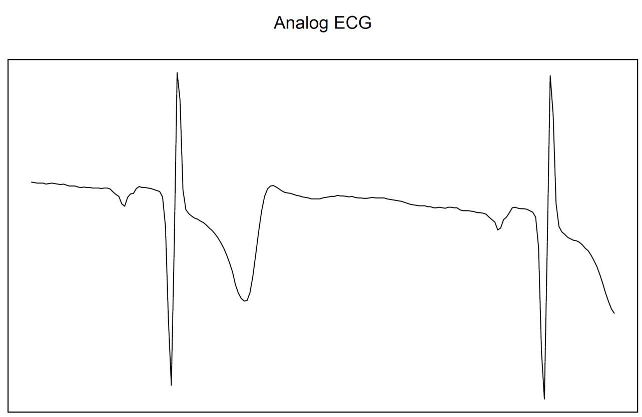

Fig. 4. ECG developed in this project

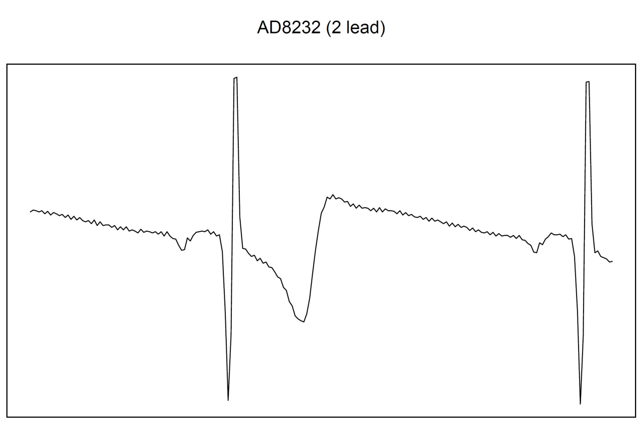

Fig. 5. AD8232 with 2 leads connected to the body

IV. DISCUSSION

A. Testing

The finished product met almost all of the target requirement specifications except from the on/off-button. This feature can easily be added by inserting a switch between the battery and the connector to the board.

It is made two testpoints between the ECG frontend and the Notch filter. This was not used because the ECG worked rigth away and no troubleshooting was required.

In the comparison between the two devices it was hard to see any difference when they was used as intended (all avilable leads connected), but when the RL-lead was detached, on the AD8232, it was clear to see that some high frequently noise showed up on the output. At the UIO electronic lab this was shown to be 50 Hz noise.

B. Further work

In the next stages of the development of the self-made ECG is to make it smaller and connect it to a smart phone via Bluetooth. I want to make the PCB small enough to fit in to a normal heart rate monitor device. In order to do that, the PCB probably have get surface mounted devices (SMD) on both side of PCB or choose a less complex device which use less space.

C. Learning points

A lot of time of this project has been spent on learning Cadstar. I hope that the experiences will bring benefit when working with other PCB design tools as well.

In the assembly process, the biggest learning point was to do the manual soldering of some components. This manual soldering can be a useful method to know when a PCB needs to change broken components.

V. CONCLUSIONS

It appears that the ECG created in this project is as good as the commercial ECG it has been tested against.

REFERENCES

[1] *TP-Stencil Printing Basics AJohnson-WWE-SM000 05-06-29*.

[2] Mahesh S Chavan, RA Agarwala, and MD Uplane. Design and implementation of digital fir equiripple notch filter on ecg signal for removal of power line interference. *WSEAS transactions on signal processing*, 4(4):221–230, 2008.

[3] Halvor StrÃžm. fys4260-2018—workshop-4—routing-and-powerplanes. 2018.

[4] J.G. Webster. *Medical Instrumentation Application and Design, 4th Edition*. Wiley Global Education, 2009.

APPENDIX A
FIRST APPENDIX

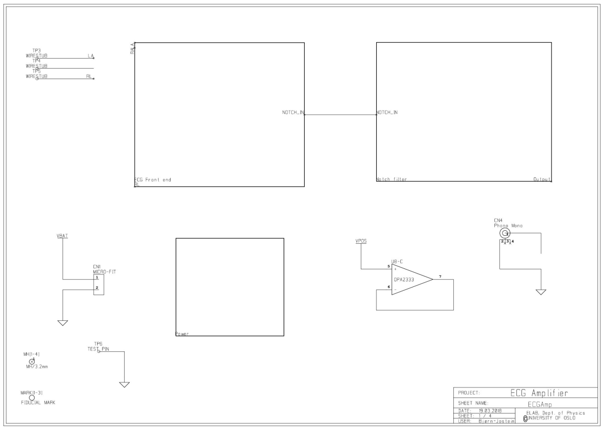

Fig. 6. Block diagram

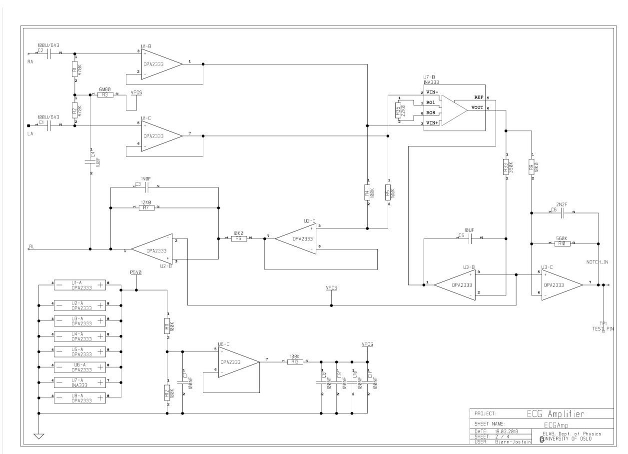

Fig. 7. Schematics ECG frontend

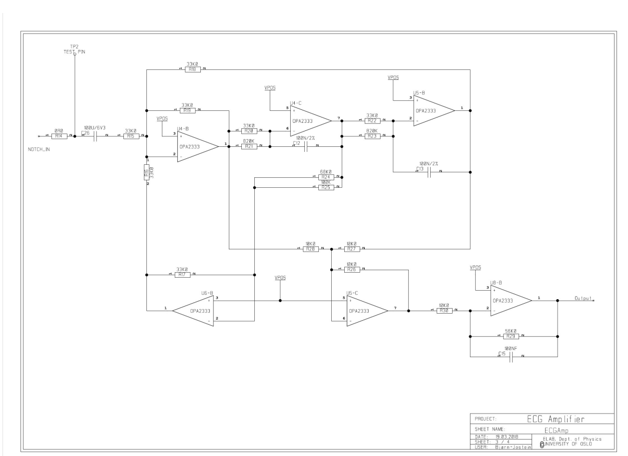

Fig. 8. Schematics Notch filter

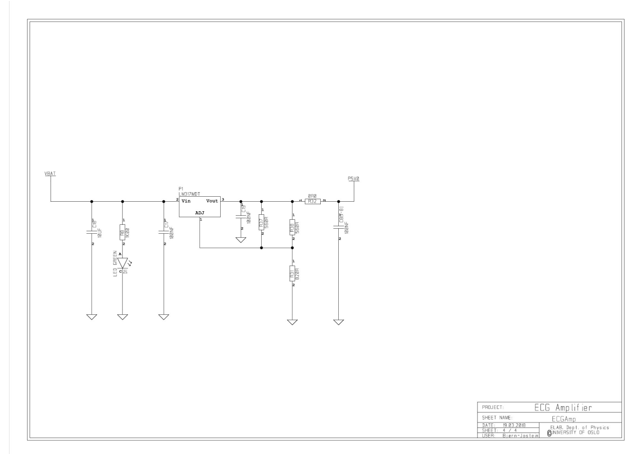

Fig. 9. Schematics Power

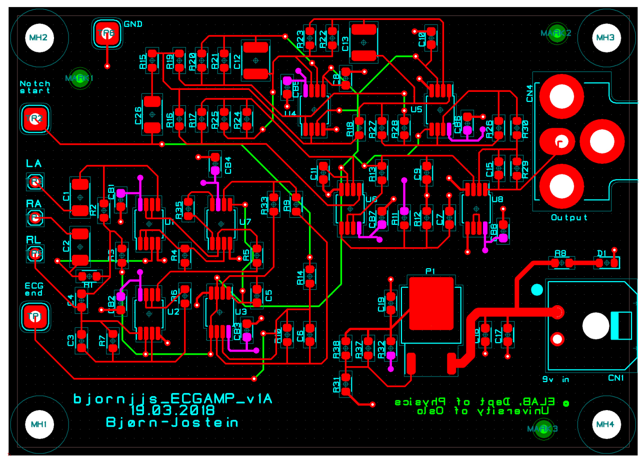

Fig. 10. PCB layout

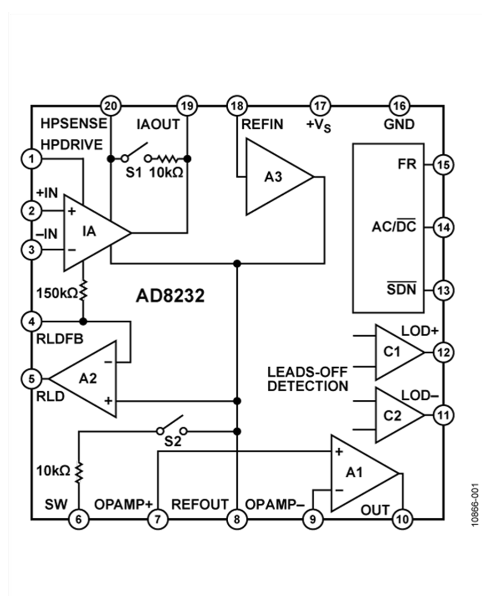

Fig. 11. Schematics AD8232