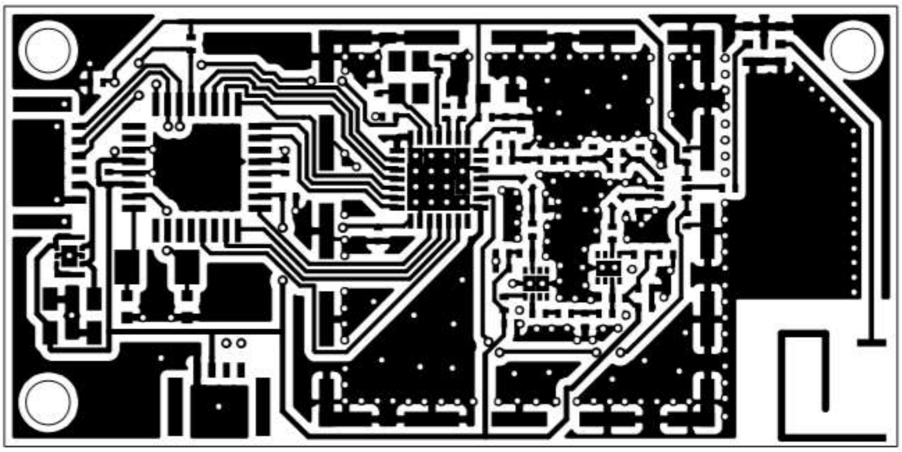
Technical University of Denmark

Mass optimization of tracking unit for stratospheric research

Hans Kristian Bech Hansen
s164025



Supervised by
René Fleron

May 16, 2022

Abstract

The following project centers around the optimization of mass of a stratospheric balloon tracker. The project starts by looking into how the old tracker functions and then investigates where and how the mass can be optimized the most. This means the project focuses on everything from the layout of the printable circuit board, the substrate of the circuit board and the use of components, to the design of the enclosure containing the tracking unit.
The project results in a mass optimization of up to 72%. However none of the created designs end up getting tested, due to a redundant mistake regarding the design of the circuit board. The project therefore still needs to be tested, in order to conclude, whether the new mass optimized tracking unit is suitable for tracking stratospheric balloons.

Contents

1 Introduction

The ability to track objects is an important aspect of modern day life. This ability is often utilized with GPS, since this gives us an exact position on the globe. We use GPS every day to either track ourselves, our car and other objects. This is utilized with either route and mapping apps like Google Maps or keep by tracking the location of our belongings with apps like ”Find my phone”. This is relatively easy, since most people own a smartphone or a computer, which makes it possible to get the data almost instantly. But sometimes we need to track faraway objects. This is especially within the fields of flight and space exploration.

DTU Space has in many years used the same type of tracker for stratospheric missions. This tracker weighs around 250 grams, which takes up a lot of the mass budget of the mission, with a a maximum of 4000 grams in the summer and 3000 grams in the winter. DTU Space has therefore a great interest in reducing the mass of the ”obligatory” balloon payload, which consists of the tracker module, the parachute and the string. It isn't easy to reduce the mass of the string and parachute, which means the tracker is the most obvious component work with.
As a result, this project was created to look into the mass reduction possibilities of the stratospheric balloon tracker, with intention to look into every aspect of the tracker. This project will therefore focus on everything from the electronic components to the physical enclosure, but with the mass in mind for all decisions.

The current tracker design was created by Andreas Rosenberg in 2016 as a variation of the Alba-tracker, which also was created by Andreas Rosenberg together with Henrik Engaard and Christian Westmark Sønnichsen back in 2015.

Figure 1: Launch of stratospheric balloon mission 18/1 - 2020. *Picture by Michalina Josephine Sigil*

2 Theory

2.1 Previous tracker design

As mentioned in the introduction, this project looks into the possibilities of optimizing the mass of an already existing tracker. The first step to so therefore to understand how the existing tracker works. A difficulty in this, is the lacking amount of available documentation from the previous project. Therefore all information is obtained by analysing the trackers Printable Circuit Board (PCB), looking into other iterations of the system and by understanding the code.

The tracker can be reduced into 5 subsystems, if the soft-/firmware is included as a subsystem. These subsystems will be described in the following subsections.

2.1.1 ATmega328P micro-controller

All subsystems are controlled using a ATmega328P based micro-controller. The ATmega328p is a micro-controller chip manufactured by ATmel. The micro-controller is build as an Arduino Pro Mini micro-controller, but with all components attached to the PCB instead of an external micro-controller. This makes it possible to program it using the Arduino Integrated Development Environment. As with the Arduino Pro Mini, it needs a FTDI programmer, in order to upload the code unto the ATmega328P chip. The micro-controller functions as the brain of the tracker. It takes the information from the GPS module and transmits it using the Long Range (LoRa) module using the SPI interface.
The ATmega328P runs on 3.3V which means it needs a power converter, in order to function properly. This power converter will be described in the ”External modules” subsection.

2.1.2 LoRa module

To transmit over large distances, the tracker contains a SX1272 LoRa chip manufactured by Semtech. The chip is surrounded by components to create a LoRa module, as a an almost direct copy of the Libelium SX1272 module for Arduino platforms. As with the micro-controller, all this exists upon the main PCB of the tracker.
The LoRa module operates at a 868 MHz frequency and has a sensitivity down to -137 dBm. This gives it, according to the manufacturer, a 21 km range when the receiver and transmitters is in line of sight. In reality the range is way above 200 km, when flying stratospheric balloons. This is obtained by applying a SAW[1] filter and 2 LNA[2]'s at the front-end. This was one of the optimizations the original project worked with.

Due to the sensibility of the SX1272, an EMI shield is needed to protect the chip from noise. The only part of the LoRa module, which not is protected, is the coaxial cable connector, which connects to the SAW filter, which connects to an antenna.

[1]Surface Acoustic Wave
[2]Low Noise Amplifier

2.1.3 External modules

The trackers contains 3 external modules, which are not a part of the main PCB: The power converter, the GPS module and the SAW filter.

The power converter sits on top of the main PCB and is attached using a piece of prototyping PCB, on which a DTUsat testboard for the power converter is attached. The converter itself is a LMZ10503 manufactured by Texas Instruments. This is a high-end step-down power converter, which can generate output voltages varying from 0.8V to 5.0V. With an output of 3.3V is it capable of achieving efficiencies above 95%[3]. This however comes at a cost of it large size, which is worsened by its configuration within the tracker.
At the input, a 3.7V Panasonic NCR18650B lithium battery is connected, which provides the power for the tracker. Due to the low temperatures of the stratosphere is this heavily isolated.

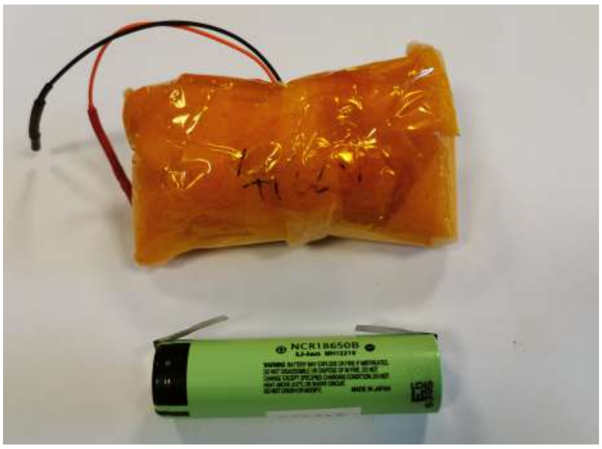


Figure 2: The isolated battery together with an un-isolated NCR18650B

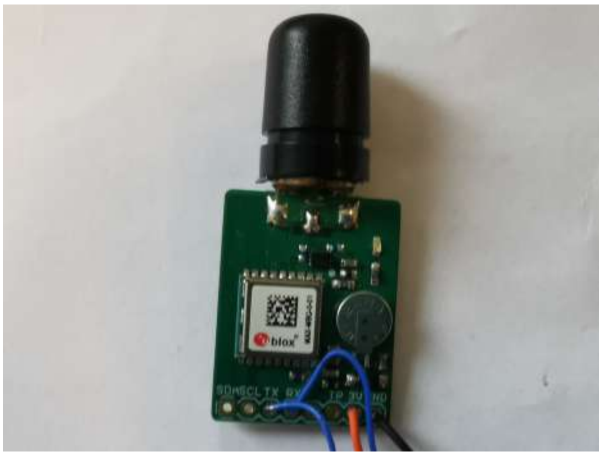


Figure 3: The GPS breakout board from Uputronics

Connected to the 3.3V output on the power converter PCB, is an Uputronics GPS module. This module is connected to the ATmega328P's serial connection, through which it transmits the trackers GPS location, which the ATmega328P transmits further to the LoRA module.

A SAW filter is located on it's own RF-PCB. It is connected with small coaxial cables and is connected between the LoRa module and the antenna. The RF-PCB isn't its own PCB, but a small piece, which is cut of from a larger PCB.

[3]Texas Instruments, LMZ10503 datasheet, p. 1

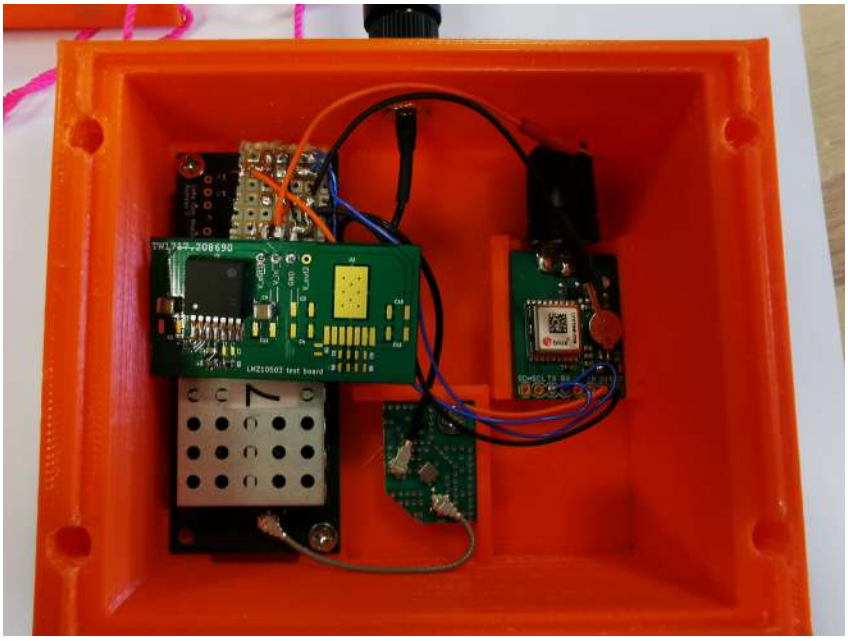


Figure 4: The inside of the old tracker. Here the SAW filter, power converter and GPS can be seen.

2.1.4 Soft-/firmware

The code which controls the tracker is written by Andreas Rosenberg. It consist of 2 different codes: One for the transmitters, which is the ones attached to the balloons and one for the receiving ground unit. The code will only be described broadly, since this project has focus on hardware and mass, but we still need to have an idea of how the tracker works.

The trackers pull the GPS data from the GPS module and transmits it every 30 seconds. This data is sent in packets, which is a string. An example of the received is shown in figure 5. Most of the figure is self explanatory, but some minor detail most

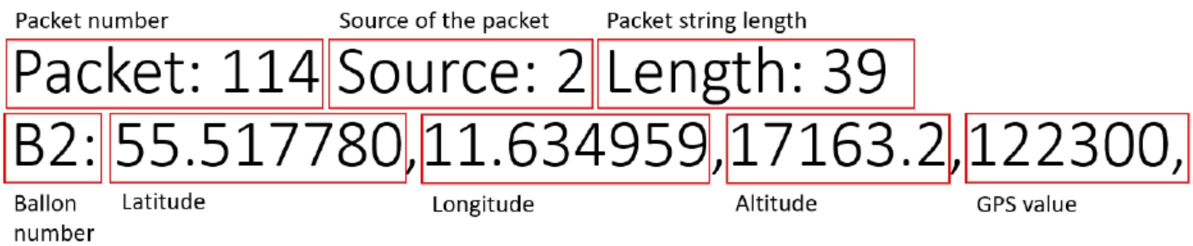


Figure 5: Example of data received from the tracker

be explained. The packet number is a number, which is used to identify the packet. For each packet sent, from the same transmitter, the number goes up. This makes it easier to look through the received data, in case the receiver gets turned off.
The source of the packet must match the balloon number. The balloon number is manually set, while the source is detected according to the transmitters address. Finally the GPS value tells us how good the data is. The higher the number, the more precise data.

2.1.5 Enclosure

Protecting the tracker is done with a rather large enclosure. The enclosure is created as a 3D printed box with a lid. On the top of the box is a small hole located, which is where the antenna sticks out. The enclosure is rather large, with a lot of wasted space, which can be seen on figure 4. Even though the battery takes up a lot of the empty space within the tracker, is is definitely not optimized.

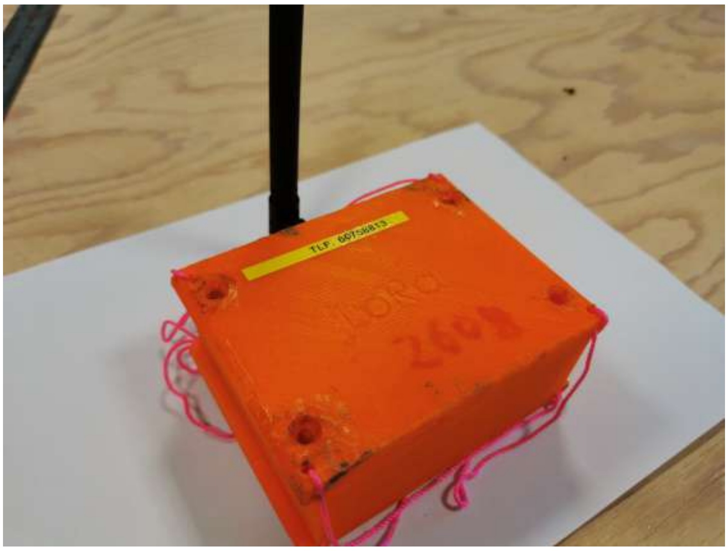


Figure 6: The enclosure of the old tracker

2.2 Transmission lines

The cobber track in a PCB, within a signal propagates, can be seen as a microstrip line, as long as the bottom of the PCB is a ground plane. An illustration of a microstrip line can be seen on figure 7. When working with radio frequencies, signals doesn't propagate like they do in simple DC-circuits. The signal propagate like a wave, and is therefore bound to the law of electromagnetics. There is therefore a lot more factors to take into account. This can be everything from the width of the electrical lines, in which the signal travels inside the PCB, to making sure the circuits impedance matches the antenna. It is crucial that these factors are met, otherwise the signal won't be at full power and some information or range will therefore be lost.

2.2.1 Impedance matching

To understand why we need to perform impedance matching, we first need to understand what impedance is. The characteristic impedance is the relation between the voltage and current along a conductor[4], and is quite similar to Ohm's law. The two

[4] Cheng; p. 432, Eq. (9-19)

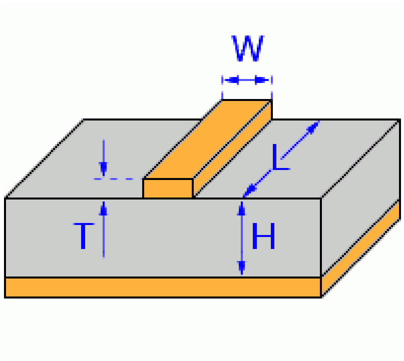


Figure 7: Illustrustration of a microstrip line (*From KiCAD's PCB calculator*)

expressions is shown below, where the similarities easily can be seen.

$$Z_0 = \frac{V(z)}{I(z)} \qquad \text{Characteristic impedance} \tag{1}$$

$$R = \frac{V}{I} \qquad \text{Ohm's law} \tag{2}$$

Impedance can be seen as the same as resistance, just for AC-circuits, while resistance is for DC-circuits. It often consists of 2 parts; a real and imaginary, since the impedance is a complex number. The resistance is the real part of the impedance, so they aren't completely different values.
An unmatched transmission line can be seen as unbalanced. If the different loads, which can be the antenna and transmission line, isn't equal in terms of impedance, the maximum power can't be reached and therefore a lot of a signal can disappear in noise. By keeping the power as high as possible, the higher is the chance of receiving the signal.

2.2.2 Line width

The impedance of a transmission lines has many factors. It involves everything from the components to the line itself. The LoRa module is impedance matched with capacitors and inductors, in order to utilize the LNA's and SAW filter. Therefore isn't this a factor we will look into, as we don't want to change any of the LoRa module. However another factor, which we need to look into, is the width of the transmission line. By either changing the substrate the line travels in or by changing it's thickness, the impedance changes and must be corrected by changing the width of the line.
The relation between the characteristic impedance and the width of a microstrip line is given by[5]:

$$Z_0 = \frac{w}{d}\sqrt{\frac{\mu}{\varepsilon}} \tag{3}$$

where *w* is the width of the line, *d* is the thickness of the substrate (corresponding to H in figure 7), μ is the permeability of the substrate and ε is the permittivity of the substrate. Equation (3) can be reformulated as:

$$w = \frac{d}{Z_0}\sqrt{\frac{\mu}{\varepsilon}} \tag{4}$$

[5]Cheng; p. 432, Eq. (9-20)

Since most datasheets for PCB substrates like FR-4 contains the dielectric constant, which is the relative permittivity to the free space permittivity ε_0, we rewrite equation (4) to:

$$w = \frac{d}{Z_0}\sqrt{\frac{\mu_r\mu_0}{\varepsilon_r\varepsilon_0}} \tag{5}$$

The relation between the permeability and the permittivity can also be described using the intrinsic impedance η:

$$\eta = \sqrt{\frac{\mu}{\varepsilon}} = \sqrt{\frac{\mu_r\mu_0}{\varepsilon_r\varepsilon_0}} = \sqrt{\frac{\mu_r}{\varepsilon_r}}\sqrt{\frac{\mu_0}{\varepsilon_0}} = \eta_r\eta_0 \tag{6}$$

To demonstrate the use of equation (5), we look at the following example:
We want to determine the width of a microstrip line on a 1.0mm thick FR-4 PCB. FR-4 has a dielectric constant of 4.5 and relative permeability of 1. At first we rewrite the equation (with use of equation (6)):

$$w = \frac{d}{Z_0}\sqrt{\frac{\mu_r\mu_0}{\varepsilon_r\varepsilon_0}} = \frac{d}{Z_0}\sqrt{\frac{\mu_r}{\varepsilon_r}}\sqrt{\frac{\mu_0}{\varepsilon_0}} = \frac{d}{Z_0}\sqrt{\frac{\mu_r}{\varepsilon_r}}\eta_0 \tag{7}$$

We can now insert our values into equation (7), since $\eta_0 \approx 337$:[6]

$$w = \frac{d}{Z_0}\sqrt{\frac{1}{\varepsilon_r}}\eta_0 = \frac{1 \cdot 10^{-3}}{50}\frac{\eta_0}{\sqrt{\varepsilon_r}} = \frac{1 \cdot 10^{-3}}{50}\frac{337}{\sqrt{4.5}} = 3.18mm \tag{8}$$

As seen by the result of (8), is this a very crude model since it's viable for all types of transmission line. This is why we in this project uses a PCB calculator for determining the width of the transmission lines.

[6] Cheng; p. 674, table B-1

3 Design and construction

The mass of a tracker can be reduced with several different adjustments. For this project the following adjustments were chosen to investigate:

1. Creation of a smaller PCB, containing all components.
2. Change of PCB substrate to something lighter than FR4.
3. Utilizing a smaller antenna for propagation.
4. Change of components into smaller ones.
5. A new, lighter enclosure design with no wasted space.

The following sections looks into each of the mentioned aspects, with focus on utilization, implementation and issues regarding each aspect.

3.1 PCB design

An easy obvious way to reduce the size of the tracking unit, is to reduce the size of the PCB. By reducing the size of the PCB, the size of the enclosure could also be reduced. Another reason to design a new PCB includes the wish to use new components and to include externally placed components onto the main PCB, which previously were connected through wires to other PCB's. This includes the SAW filter, the GPS module and the power converter. The extra wiring and PCB's adds unnecessary mass, which easily can be removed, by adding the components to the main PCB. In some cases will this of course make the main PCB larger, but the total mass will still be reduced, by removing the extra wiring.

The main challenge of creating a new layout, was the nonexistence of the PCB layout of the old tracker. The design was obviously created at some point, but was never uploaded to the DTUsat GitLab. However another version of the tracker was found on the GitLab. This version was created by Denis Tcherniak and was modified to fit into the DTUsat-3 system. This includes a stack connector, another antenna connector and a larger and more complex STM32 microprocessor. The base design for the LoRa module and the SX1272 chip was still the same as the original tracker, which meant this design could be duplicated for the new tracker.

Using the schematic from the DTUsat version of the tracker and by looking at the old tracker, was it possible to recreate the original tracker, with a slight reduction in size. The redesign of the tracker also made it possible to add new components, which all will be described later. In the end, a total of 4 new tracker designs were created. These will be described in detail in the ”Final Designs” section, while this section and the following sections will focus on the overall design.
The methods and tools to create the new PCB's is described in the following subsections.

3.1.1 Tools and software

There is a lot of software, which is suitable for designing PCB's. These programs range from being free, open-source software to expensive, high-end software, with a price of several thousand DKK. These programs also differ in user friendliness, as the expensive programs tend to be more user friendly, since they apply to a market and need to turn a profit.

The choice of software fell upon KiCAD, which is a an open-source EDA (Electronic Design Automation). KiCAD is widely supported by everything from open-source platforms like Arduino and Raspberry Pi, to distributors like Digikey.com. It's even supported by some universities and even CERN, the research facility. Almost every PCB manufacturer supports KiCAD files for manufacturing PCB's, which also makes KiCAD a great choice. The real reason for the choice, is because of its use in DTUsat projects, as the modified tracker for DTUsat-3. It was therefore the obvious choice, since future projects probably will look into optimizing this project and KiCAD being the standard for DTUsat projects.

With KiCAD being an open-source software, the program has a steep learning curve. Because of this we chose to design the trackers main PCB from scratch, even though a the DTUsat-3 version already existed. Another reason for this, was the learning experience. Starting on an already existing design can be difficult, when you have no experience with the software and all tutorials focus on starting from scratch. Due to this, the design of the PCB took nearly a week. But it resulted in a more optimized design, which meant a lighter PCB. The new designs will be described in the "Final Designs" section.

3.1.2 Transmission line width

The signals between the SX1272 LoRa chip and the antenna operates at a frequency of 868 MHz. The cobber tracks in the PCB, in which this signal travels, therefore needed to be optimized for this frequency. To make sure all factors were taking into account, we used KiCAD's built-in PCB calculator. This gives us a much more precise width, since this takes all factors into account. We assumed our FR-4 PCB's would be 0.5mm, since the original tracker PCB was 0.5mm and used the calculator to get a result of 0.356mm, which we applied to all RF lines. All other lines were set to 0.3mm for convenience and to reduce the risk of having a gap in a line, which can occur with more narrow lines.

3.1.3 SAW filter placement

As mentioned, one of the ways to reduce mass of the tracker, was to optimize the mass by placing the SAW filter on the main PCB, instead of its own little piece of RF-PCB. The SAW filter is the last component, which the 868 MHz signal passes through, before it reaches the antenna. It is therefore almost placed at the exact same place in all 4 designs.
The placement of the SAW filter is right outside the EMI shield. This is due to the lack of space underneath the EMI shield. This placement works out, since the SAW filter

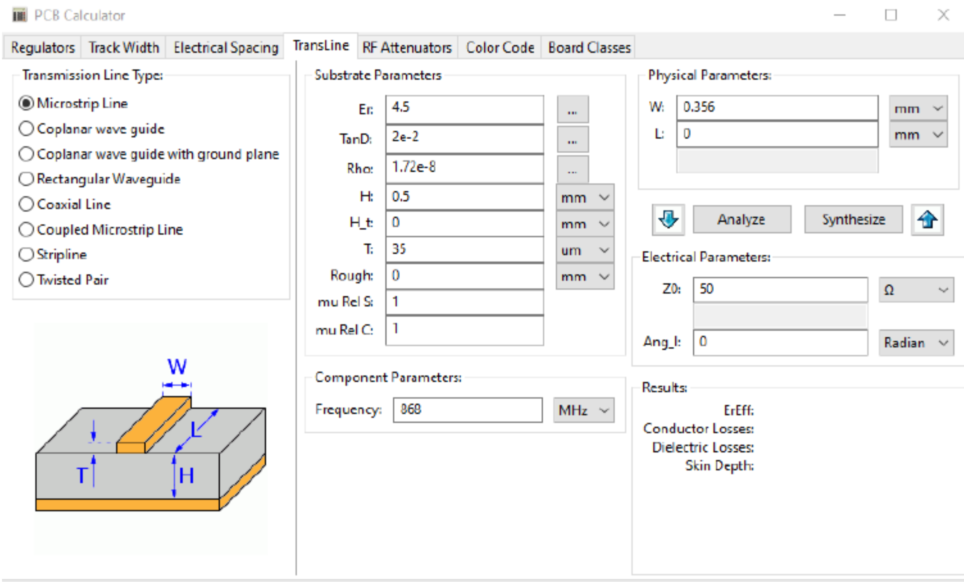


Figure 8: Screenshot with the used information

isn't as sensitive to noise as the SX1272 chip.

By placing the SAW filter upon the main PCB, the mass from the SAW filter is reduced from 4.4g, which is the external PCB and wiring, to below 0.1g.

3.2 PCB substrate

To find and chose a PCB substrate other than FR-4 may seem as an odd choice, since FR-4 is the standard for a reason and is a quite light material. For an example weighs the PCB of the old tracker only 4.7 grams. But it was still interesting to investigate, if this number could be reduced to something even lighter. A lot of work was therefore put into finding a new suitable substrate, which would be lighter, without affecting the performance to much. This was also one of the few "new" technologies which this project investigated and was therefore especially interesting because of its possibilities.

Substrates lighter than FR-4, which is also suited for circuit boards, are very limited and the first choice fell upon an already acquired piece of kapton laminate. This substrate was made by C.I.F. an consisted of polyimide(kapton) film with cobber foil on one side. To make a PCB out of this substrate, we tried to create it using DTU Space's in house production facilities. This meant the use of a photo positive aerosol spray, which needed to be applied in a dark room, to avoid UV-lighting, which will vaporize the substrate. After a night of drying the substrate was put into a UV light chamber together with a transparent paper, which had the circuit printed upon(see figure 9). The transparent paper were placed on top of the cobber layer. The UV light vaporized the aerosol, with exception of the printed circuit. After this the substrate were placed in acid, which would remove all cobber, which wasn't covered by the aerosol. However the cobber foil on the substrate were to thin, which resulted in all the cobber dissolving in the acid. We therefore needed another way to produce a flex print PCB.

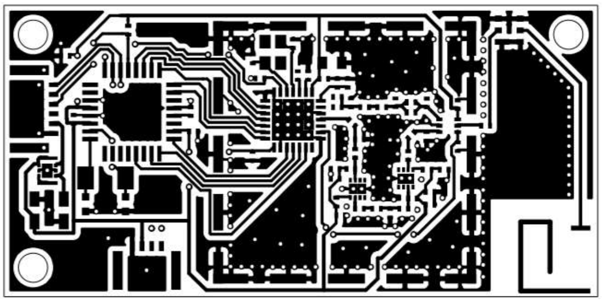


Figure 9: The circuit printed upon the transparent paper

Due to the complications of producing a kapton based PCB, we chose to order a flex print PCB from the manufacturer OSH Park. This 2-sided flex print ended up weighing only 1.0g, and is therefore even lighter than FR-4 PCB. The PCB substrate is made by Panasonic and has the designation Felios R-F775.
Due to the lack of information about the materials relative dielectric constant, we chose to keep the same track widths as the FR-4 based trackers.

3.3 Antenna

A large part of the trackers mass comes from it's antenna. It was therefore ideal to look into the use of a smaller antenna. Inspired by the GPS module, which will be described in one of the following subsections, the choice fell upon a SMD chip antenna. By utilizing a chip antenna, the antenna itself could be placed upon the main PCB and therefore completely avoid the use of small coaxial cables.

The chosen antenna is a Johanson Technology 868 MHz ceramic chip antenna. This choice is based on the antennas frequency range of 858-878 MHz, its impedance of 50 Ω and its small form factor. The only issue with the antenna, is its need for a large ground plane surrounding the antenna.[7] By requiring a large ground plane, the antenna calls for a larger PCB. The different solutions to this problem can be found in the "Final designs" subsection, as Design 1 and 2.

By applying the ceramic chip antenna, the mass of the antenna went from 22.8g to under 0.1g.

3.4 Power converter

The power converter was another critical aspect of the tracker, which needed to the changed. The previous trackers power converter was large and bulky and the whole setup with a prototyping PCB was unsustainable. The new power converter needed to be light, possible to mount on the main PCB and efficient. The efficiency is directly linked to the size of the battery. The less efficient the power converter is, the more power is needed to complete a flight mission. And batteries are heavy. Therefore: An efficient power converter makes opportunity for a smaller battery solution, which leads to a lighter tracker.

[7] Johanson Technology, 868 MHz antenna datasheet, p.3

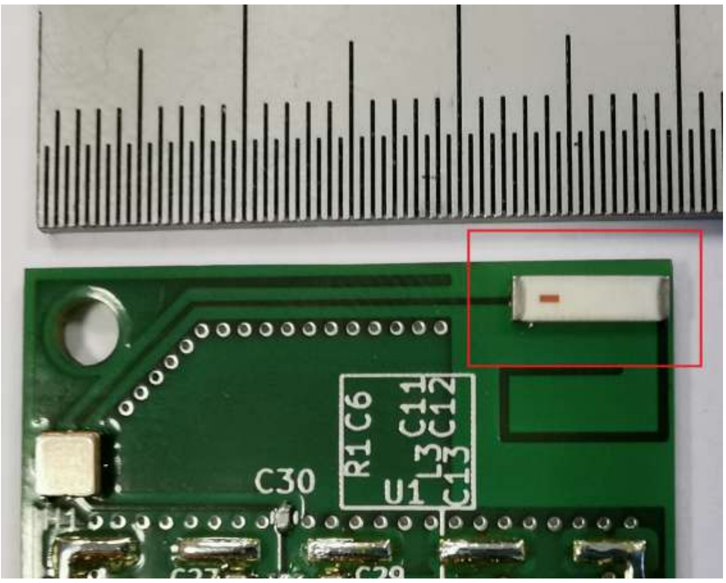


Figure 10: 868 MHz chip antenna mounted upon one of the final designs

As the ATmega328P and the SX1272 uses 3.3V and the existing battery provided 3.7-4.2V, a down-converter was needed. The choice fell upon the XCL205 Step-Down micro DC/DC Converter from Torex Semiconductor. This power converter IC[8] has the small size of 2.0x2.5x1.0mm and was therefore perfect for this project, which required as small components as possible. It also has a high efficiency of 90%, when using a 4.2V input source.[9] With its small form factor and high efficiency was the XCL205 the obvious choice. But the XCL205 still needed some extra components, in order to be configured correctly. This included a 10μF and a 4.7μF capacitor, which needed to be configured according to figure 12.

By utilizing the XCL205 power converter, the mass of the power converter was reduced from 4.4g to under 0.1g.

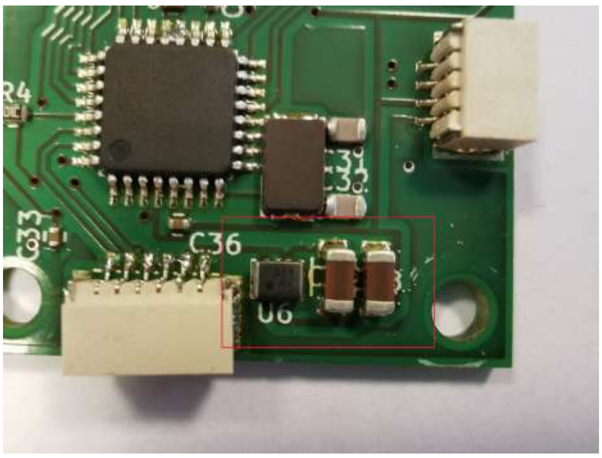


Figure 11: XCL205 placed together with its configuration capacitors

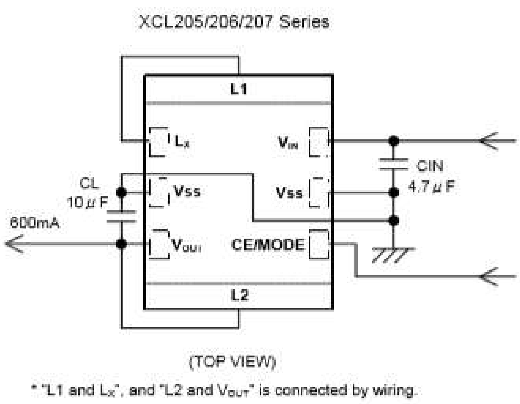


Figure 12: Configuration of the XCL205

[8]Integrated Circuit

[9]Torex Semiconductor, XCL205/XCL206/XCL207 datasheet, p.1

3.5 Battery solution

When looking into the use of a new power converter, the idea of a new battery solution came up. The old tracker used a Panasonic NCR18650B, which had a large capacity but wasn't optimized for cold conditions. This meant the battery needed isolation, in order to function properly in the -50°C to -40°C cold stratosphere. It was therefore ideally to find a battery solution, which could work in the cold conditions. The choice fell upon using 3 Energizer Ultimate Lithium AAA batteries. These batteries can withstand down to -40°C, which makes them suitable for stratospheric flight. Each AAA battery deliver 1.5V, which is why 3 is needed. By connecting them in series, a voltage of 4.5V was obtained. The 3 batteries is placed in a battery box, which connects the batteries in series.

Using the 3 AAA Lithium-ion batteries as a power source, the weight from the battery solution was reduced from 50.4g to 36.7g.

Figure 13: Battery box with batteries

3.6 GPS module

For reducing the mass of the GPS module, a number of options were available. The first and most obvious, was to find a smaller GPS breakout board. The other was to incorporate a GPS on the main PCB itself. Even though placing all components on the main PCB was listed as a possible way to decrease mass, the choice fell upon just using a smaller breakout board. This choice is based on the lightness of the uBLOX MAX-M8C Pico Breakout with Chip Antenna from Uputronics, the same manufacturer of the original GPS. This small GPS only weighs 1.5g, due to its use of a chip antenna and a thin, light FR-4 PCB. Even by utilizing the same components upon the main PCB, the main PCB's extra size would result in a mass over 1.5g, since the PCB would be thicker. By using a breakout board, it would also be easier to change the GPS, in case it would either break or not perform well enough.
By using a breakout board, the main board needed an extra connector for the GPS. This didn't increase the size of the main PCB, due to some unused space. The used connector is described in the next subsection.

By using the uBLOX MAX-M8C Pico Breakout with Chip Antenna instead of the original GPS breakout board, the mass of the GPS was reduced from 11.1g to 1.5g.

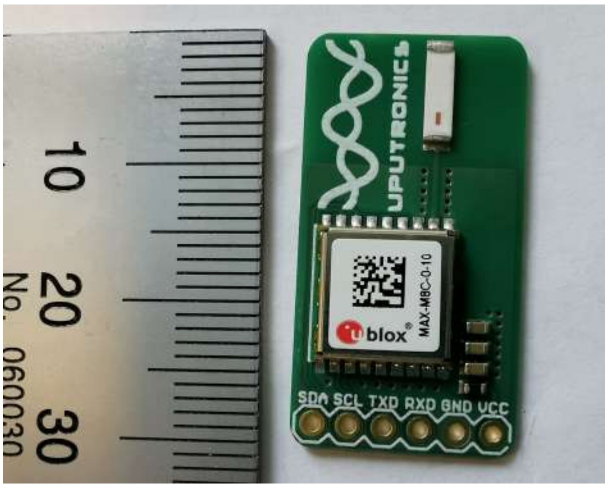


Figure 14: The uBLOX MAX-M8C Pico Breakout with Chip Antenna

3.7 Connector

Even though it doesn't add much weight, the connector was also looked into. The previous tracker used 5 pins on top of the board, onto which the power converter prototype board was connected. This both gave the tracker a weird geometrical shape and a bit unstable setup. Therefore a new connector seemed like a good idea.
The new connector needed to be light, have a minimum of 5 pins, for connecting the FTDI for programming, and to be strong in its connection, while still be relatively easy to disconnect. The choice fell upon the Qwiic Connect System from SparkFun, which is a variant of the JST connector. The connector comes in 3 variants: 4 pins, 6, pins and 8 pins. Since we needed at least 5 pins, the 6 pin version were chosen for the FTDI/battery connection, while the 4 pin was chosen for the GPS connection.

With the Qwiic connector the mass was reduced from 0.3g to 0.1g. This doesn't seem like much, but the change of connector also resulted in lighter wires, which also had a small impact on the total mass.

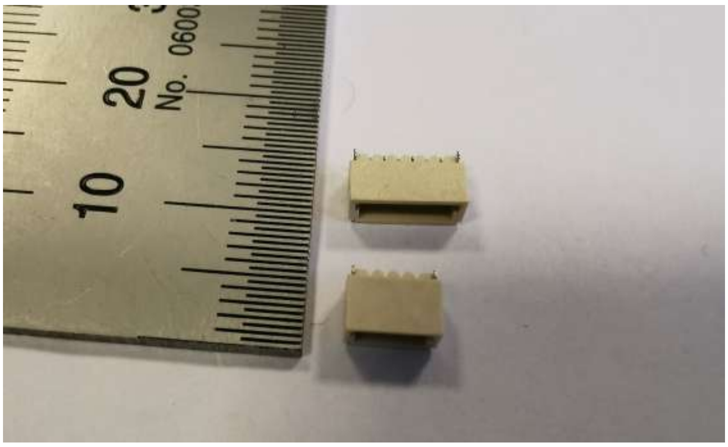


Figure 15: Qwiic connectors. *Top:* 6-pin for power and programming. *Bottom:* 4-pin for GPS

3.8 Enclosure

With all the components selected, the tracker needed an enclosure. Most of the old trackers mass, came from its large enclosure. This were therefore one of the key features to optimize. Just like with the old enclosure, the new enclosure was created with PLA[10] using a 3D printer. It therefore needed to be designed using a 3D CAD[11] software. For this project the program SolidWorks were utilized.

When designing an enclosure, many factors needs to be taking into account. This includes the size of the PCB, the height of the EMI shield, the placement and type of antenna etc. Also the placement of the connectors, the GPS module and battery pack is quite important. Each important aspect is described in the following subsections. However the overall design is more or less the same. It is split in three parts: A main enclosure, a lid for covering the main PCB and a lid for covering the GPS, which all is displayed in figure 16.

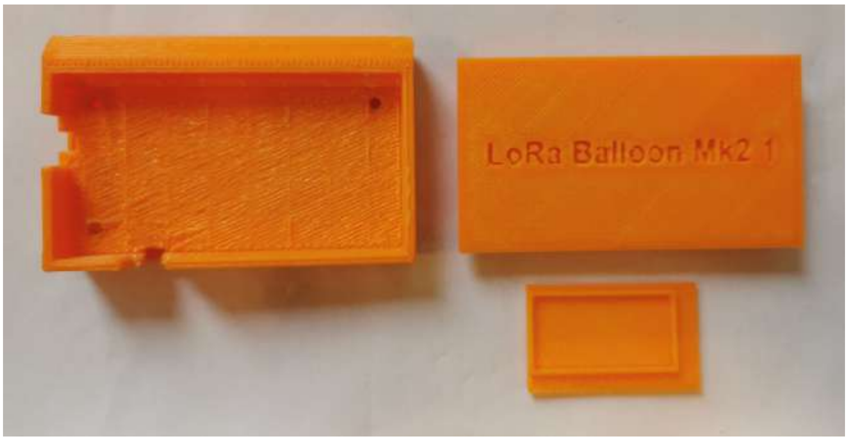


Figure 16: The enclosure for Design 1, which is more or less the standard for all designs

3.8.1 Main board placement

At the center of the enclosure is the main PCB placed. All other aspects is created around this, since it is the heart of the tracker. The main PCB is enclosed in a box, with mounting holes, so the tracker can be attached to the enclosure. The box is build in two parts: A box and a lid, which closes the box. The box has 2 holes. One for power and programming and one for the GPS. The placement of the main PCB can be seen on figure 17.
The size of the tracker differs between the final tracker designs. Two of the designs also contains a hole for the antenna cable. This will be described in detail in the "Final Designs" section.

[10]Polyactic Acid - a thermoplastic alphatic polyester used for 3D printing
[11]Computer Aided Design

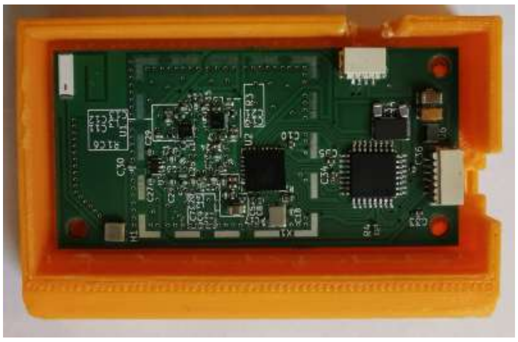


Figure 17: The placement of the PCB

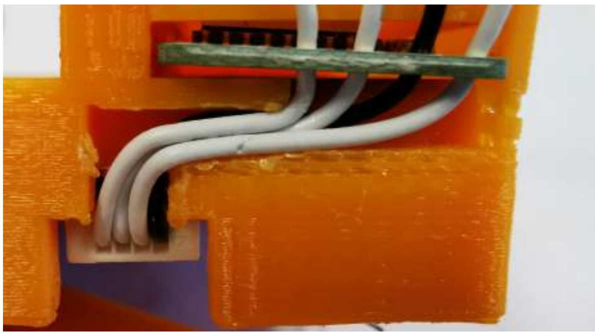


Figure 18: The notch for the GPS wiring

3.8.2 GPS mount and wiring

The GPS module is placed on the backside of the tracker. Here it has its own small slot, which also has its own lid. This slot is 31.0x16.5mm, which is just enough to hold the GPS in place. The side of the tracker, between the GPS slot and one of the holes for connectors, contains small notches for wiring. This makes it possible to avoid lose wires, which can be torn off or disconnected during flight. The notches can be seen in figure 18.

3.8.3 Battery pack placement

As mentioned in the selection of the battery solution, the batteries are contained in a box with room for 3 AAA batteries. This box is placed on the back of the enclosure, where it is mounted with double sided tape. The wiring of the battery box is placed toward the GPS slot, where a cable guide is located. This guide provides at stable and protected guideline for the power cables, which goes all the way to the power connector on the main board. The side of enclosure contains notches for wiring, as with the GPS. The cable guides and the placement of the battery box can be seen on figure 19.

Figure 19: Battery box placed on the back of the enclosure. The guides follow the edge of the GPS enclosure.

3.8.4 Stratospheric balloon mount

The tracker is created for stratospheric research and will be mounted to a balloon train, connected to a stratospheric balloon. It therefore needs a solid and easy way to mount the enclosure to a string. This is done by making a cylinder on the side of the tracker, which the string can go through. At each end of the cylinder is two holes located. By passing the string through one of the holes and back through the cylinder, then through the hole in the other end, the string firmly secures the tracker. This can be seen on figure 20.

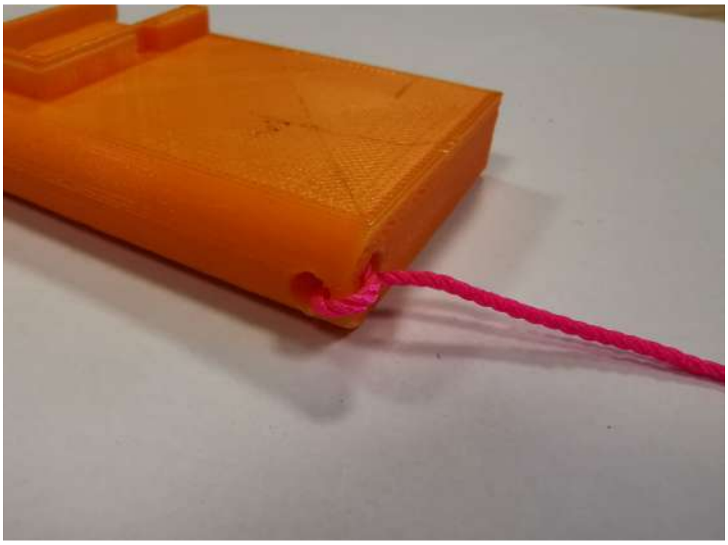


Figure 20: Holes with string for mounting the tracker to a stratospheric balloon

3.8.5 Water resistance

So far the enclosure consists of a number of sections, which either contains a lid or is open. The enclosure is therefore not water resistant. This is a problem, because of the atmosphere, which often contains rain. The whole enclosure is therefore covered in kapton tape. This is a thin, yet strong form of tape, which also is waterproof. By covering everything in tape, we make sure everything is held in place, while covering all cracks and notches. This makes the tracker water resistant, but not waterproof. If the tracker ends up in the ocean, is it just a matter of time, before the water gets to the components and causes a short circuit.

3.9 Other, discarded changes

Some other ideas for optimizing the mass of the tracker were proposed, but ended up being discarded. These are listed below with a small description.

- **Other microprocessor**
  Some work were put into looking for a smaller microprocessor, to replace the ATmega328P. The only Arduino compatible processor, which were smaller than the ATmega328P, was the ATtiny85. However this didn't support the serial communication needed for the GPS or enough memory for the GPS libary.

- **Power optimization of software**
  A way to save a lot of mass, is by reducing the size of the battery solution. A way to save power, is to look into the soft-/firmware of the tracker and change it for lowering the power consumption. This could for example be done by increasing the time between pulses. This would make the stratospheric balloons harder to track, which is why we didn't do it. Other ways of power optimization in the code could probably be done, but due to the codes complexity and sparse commenting, this wasn't utilized.

- **Self-made power converter**
  When we looked into changing the power converter, we started looking into how to create one ourselves. However after the discovery of the XCL205 this plan were discarded, since we never would be able to create something´as small as the XCL205.

4 Final Designs

To test which effect the different parameters had on the trackers performance, 4 different tracker designs were created. Each tracker were designed to see which parameters affected the performance, with the end goal of testing all 4 on the same stratospheric balloon, together with an existing, functioning tracker.
Most of the trackers hardware is identical. The only difference is the antennas, antenna placements, PCB substrate and the enclosure. These are the aspects each section will focus on, since the general design and choice of components has been described in the previous section.

4.1 Design 1 - FR-4 board with chip antenna on the top

4.1.1 PCB

Figure 21: PCB of the first design

The first design has a FR-4 PCB and is using the ceramic chip antenna. The PCB measures 30x61mm, compared to old tracker which were 31x75mm. The new PCB is smaller than the original trackers PCB, even though it includes a power converter, SAW filter, antenna and GPS connector, which all were separate on the old tracker. The PCB is smaller than the previous trackers PCB, but ended up heavier. This is because the PCB is thicker. The reason for this thicker PCB, is the choice of manufacturer. To get the fastest delivery and the cheapest print, we used the manufacturer Aisler. Aisler focuses on making it easy and cheap to order and manufactor prototype PCB's. Therefore isn't it possible to select PCB thickness, which resulted in a 1.7mm thick PCB. The PCB itself weighs 6.1g, which is 1.4g heavier than the original.

4.1.2 Antenna

The antenna is placed on the top of the PCB, together with all the other components. The ground plane may seem to small, compared to the recommended configurations in the datasheet[12], as the ground plane only surrounds the feed line. However this design was created to test, if the EMI-shield can function as a ground plane for the antenna. It also simplifies the soldering of the PCB, if there is no components on the backside of the PCB.

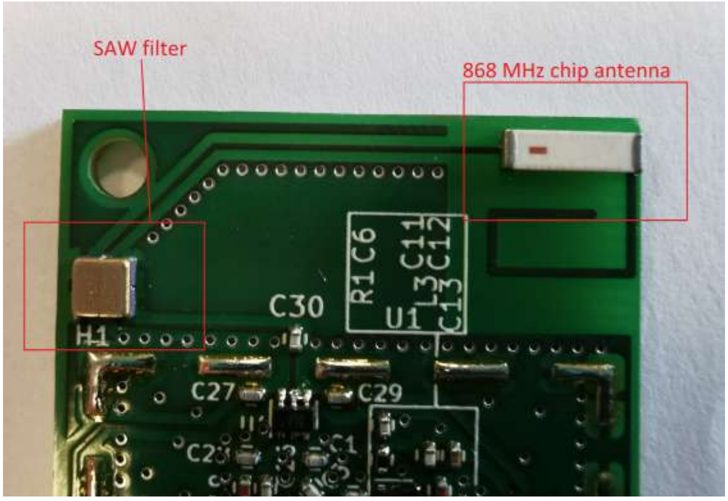


Figure 22: The SAW filter and antenna configuration of Design 1

4.1.3 Enclosure

The enclosure of the first design is, as mentioned before, based around the placement of the PCB. This design is basically the exact described enclosure, from the previous section, since it were the first design and served as a template for the other designs. With the lid attached, the enclosure protects the PCB with no other holes than the connectors. To optimize mass, almost no extra space is present inside the tracker. Almost all walls are a total of 4mm thick. This is maybe a bit excessive, but it makes sure the enclosure doesn't break to easily.
The enclosure weighs 25.6g, which is remarkably lighter than the old trackers enclosure, which weighed 156g.

4.2 Design 2 - FR-4 board with chip antenna on the bottom

4.2.1 PCB

The second design also has a FR-4 PCB and utilizes the 868 MHz chip antenna. It has the exact same measurements as the first design, measuring 30x61mm. This PCB was also ordered from Aisler and is therefore also 1.7mm thick and also weighs 6.1g.

[12]Johanson Technology 868 MHz antenna datasheet, p.3

4.2.2 Antenna

The only difference between the first and second design, is the placement of the chip antenna. On the second design is the antenna placed on the bottom of the PCB. Here it can utilize the complete ground plane of the bottom, which fills the full area of the PCB's backside. This should, in theory, provide the best efficiency of the antenna.
It is a bit harder to solder, since it's placed opposite all other components. Luckily is the antenna easy to solder using a soldering iron, so this won't be an issue.

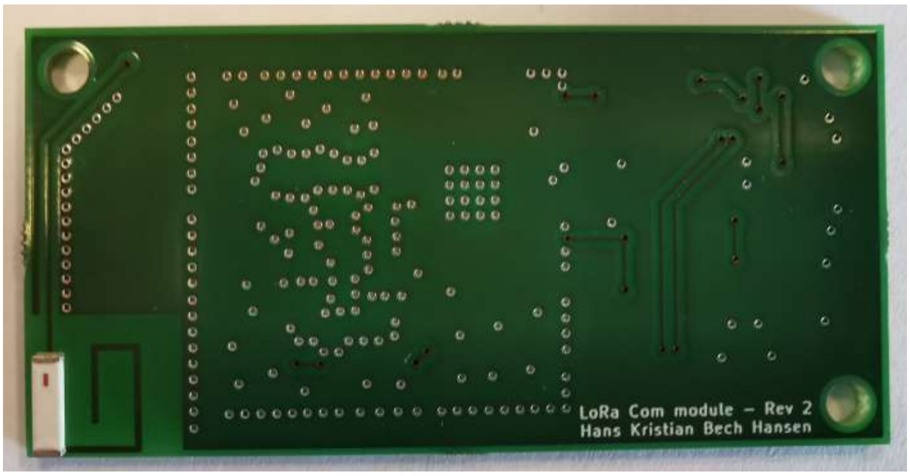


Figure 23: The antenna configuration of Design 2, with the antenna mounted on the back of the PCB

4.2.3 Enclosure

The enclosure of the second design is almost identical to the enclosure of the first design. However due to the chip antenna placed on the backside of the PCB, a small space for the antenna has been created. The small space of the enclosure can be seen on figure 24. This enclosure weighs 26.1g and is therefore the lightest enclosure.

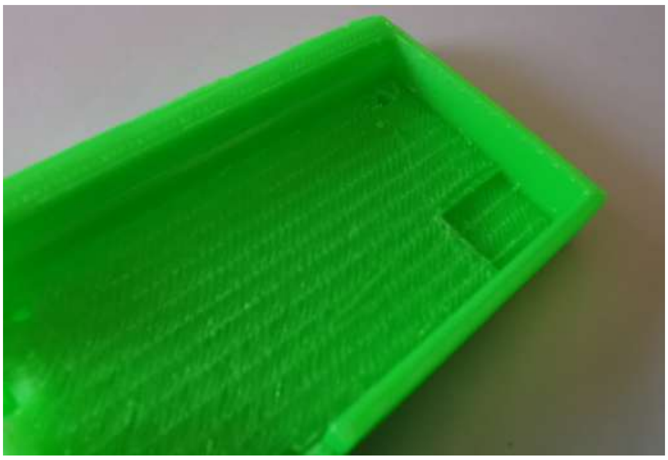


Figure 24: The small antenna space in Design 2's enclosure

4.3 Design 3 - FR-4 board with coaxial connector

4.3.1 PCB

The third design also has a PCB made of FR-4, but has a coaxial connector instead of the chip antenna. Due to the use of a connector instead of the chip antenna, the PCB is a bit smaller than Design 1 and 2's PCB and measures 30x56mm. As this also is manufactured by Aisler, it is also 1.7mm thick. This gives the PCB a mass of 5.6g, which is still heavier than the original trackers PCB.

4.3.2 Antenna

As mentioned has this design a coaxial connector instead of the chip antenna. By using this connector, a regular antenna, like the one on the old tracker, is utilized. This design is therefore almost identical to the original, with the exception of being smaller and having the SAW filter on the main PCB. It serves as a test to see, if the SAW filters placement on regular FR-4 impacts the performance.

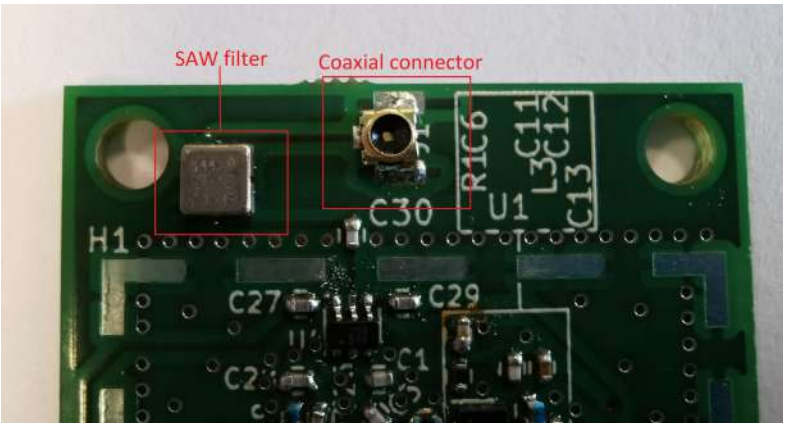


Figure 25: Antenna configuration of Design 3 with a coaxial connector.

4.3.3 Enclosure

The enclosure of the third design is a bit smaller than the enclosure for the two previous design. This is due to the smaller PCB. However it is still heavier than the other enclosures, with a weight of 28.2g, because of its mount for an antenna. It also has a hole for a cable to connect the antenna to the PCB.

Figure 26: Antenna mount and cable hole for Design 3 and 4

4.4 Design 4 - Flex print with coaxial connector

4.4.1 PCB

The fourth and final design has a PCB made from flex print and also uses the coaxial connector. The PCB has the same measurements as Design 3, which measures 30x56mm. However it is much lighter. This is due to the PCB substrate being the Panasonic Felios R-F775. The PCB therefore only weighs 1.0g, which is almost a fifth of the original PCB's mass.

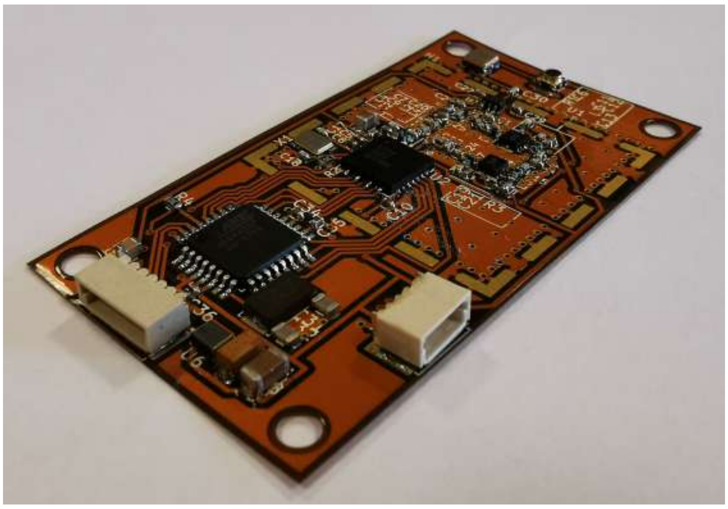


Figure 27: Design 4 PCB made out of flex print

4.4.2 Antenna

The flex print design has a coaxial connector for connecting a regular antenna. A regular antenna is utilized, instead of the 868 MHz chip antenna, to test how well the flex print handles the RF signal. If the flex print also had a chip antenna, more variables would affect the performance, which would make it difficult to determine the impact of the flex print PCB. This results in the design being heavier than Design 1 and 2, despite the lighter PCB. However it is important for testing the performance of the flex print.

Figure 28: Enclosure with antenna for Design 3 and 4

4.4.3 Enclosure

The flex print design uses the same enclosure as the third design. The print has the same dimensions and therefore isn't there any reason to change the design of the enclosure.

4.5 Design problems and issues

Even though 4 designs were created, one redundant mistake ended up making all 4 tracker designs non-functioning. Due to misunderstanding of the datasheets, when creating footprints[13] for the XCL205 power converter and the SAW filter, both of these footprints ended up wrong. Both footprints ended up mirrored, since the footprints in the datasheets shows the buttom view, and not the top view. The mistake occurred due other datasheets using top view, as with the ATmega328P's datasheet. This standard mistake results in the whole unit not getting power, since the XCL205 power converter is wrongly configured, and therefore not producing 3.3V for the rest of the system. The SAW filter is still usable, since it's configuration makes it possible to flip, due to it only having an input and output.

The flaw of the trackers were noticed to late to order new PCB's and therefore none of the trackers was usable. However a corrected version of the PCB design has been created and will be uploaded to the DTUsat GitLab for future projects.

[13]Footprints marks the placement of the pads when designing PCB's

Another problem, which occurred during the assembly of the trackers, was the soldering. Due to the small size of the components and some of the IC's only having pins on the bottom, it needed to be soldered using solder paste. To do this a solder paste dispenser was used. However this dispenser would, even on its lowest setting, dispense to much solder paste. When the PCB's then were put in the oven, the components would ”float around”, due to the excess solder paste.
This issue was fixed by ordering stencils. A stencil is a thin metal plate, which has the solder pads cut out (see figure 29). When placing the stencil directly upon the PCB's, a thin layer of solder paste can be applied and scraped of, so there only is the thickness of the plate left. By using a 100 micrometer thick stencil, an almost perfect layer was applied, which resulted in fewer corrections. The difference between soldering with and without a stencil can be seen on figure 30 and 31.

By using solder paste and a soldering oven, another problem occurred. When the thin flex print PCB was in the oven, it bent, which meant not all pins of the IC's were touching the pads when the solder cooled an clotted. A solution to this could be a metal block, onto which the PCB could be mounted using the screw holes. However this wasn't tried, since the PCB was faulty and we didn't want to waste components.

Figure 29: Stencil used for Design 1

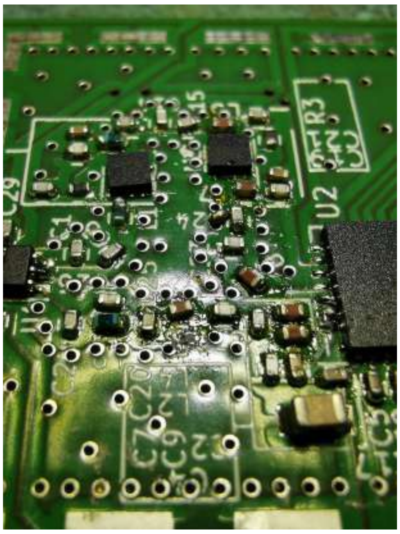


Figure 30: Result of soldering without stencil

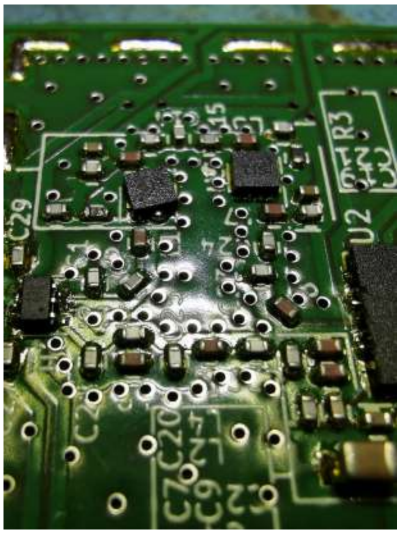


Figure 31: Result of soldering with stencil

5 Testing

Even though no trackers were operational and no tests could be performed, a test setup was still created. One of the goals of the project was for the new, lighter trackers to operate just as well as the original. It is therefore obvious to do a comparison test. This test can be performed by attaching the original tracker to a stratospheric balloon together with the 4 newly designed trackers. Then the data from all 5 trackers can be compared, since they all would be at the same position during flight, with just a very small difference in elevation. After the mission, the data can be plotted and the original trackers position can be utilized as a baseline, which the new trackers can be compared to.

The data from Design 1 and 2 can be used to determine, if the chip antenna functions properly and to conclude which side of the main PCB that has the best ground plane.
Design 3 has the almost the same function as the old tracker, as this design is the closest to the old design. By looking at the data from Design 3, it can be determined if the SAW filter functions properly when mounted on the main board, instead of having it's own separate RF-PCB. The data from the fourth design is maybe the most interesting. This data will show if the flex print can be utilized for LoRa communication and therefore if the use of flex print can be used to optimize the mass of the tracker even further.

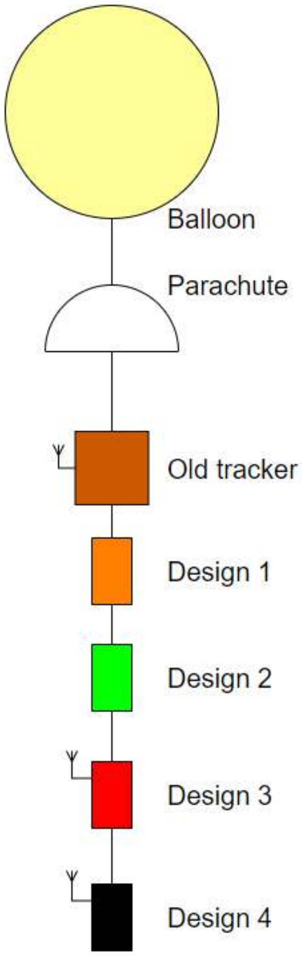


Figure 32: Proposed test setup

6 Results and comparison

Due to the lack of testing, the only results to compare is the resulting mass of the trackers. This is also the focus of the projects, but ideally the performance of the trackers could also be compared. A comparison of the masses of the final designs can be seen in table 1. When a mass is set to 0.0g, it means it is accounted for elsewhere. For an example is the power converter moved to the main PCB for the new designs and is therefore accounted for in the mass of the main PCB.

Part / Tracker	**Old tracker**	**Design 1**	**Design 2**	**Design 3**	**Design 4**
PCB (with components)	8.2g	8.9g	8.9g	8.0g	3.4g
GPS	8.6g	1.5g	1.5g	1.5g	1.5g
Antenna	17.6g	0.0g	0.0g	17.6	17.6g
SAW filter	1,0g	0.0g	0.0g	0.0g	0.0g
Battery	50,7g	36.7g	36.7g	36.7g	36.7g
Enclosure	165g	26.6g	26.1g	28.2g	28.2g
Total mass	260g	73.7g	73.2g	92.0g	87.4g

Table 1: Final mass of the designs

When looking at the total mass of the final designs, is it possible to compare them to the old tracker. When we do this, we get the following reduction in mass for each design:

Design	**Reduction in mass [g]**	**In %**
Design 1	186.3	$\approx$72%
Design 2	186.8	$\approx$72%
Design 3	168.0	$\approx$65%
Design 4	172.0	$\approx$66%

Table 2: The final mass reduction for each design

7 Conclusion

Despite the fact that no of the newly designed trackers were functional, some results were still obtained. Namely the mass reduction, which also were the focus of the project. The trackers mass were reduced with 65% to 72%, depending on design, which results in trackers varying from 73.2g to 92.0g. When these trackers are completed and tested, can that number probably be reduced even more. This is of course only if the flex print tracker and chip antenna works as well as the old tracker. Even if neither the chip antenna nor the flex print ends up with good results, the projects would still be considered a success. With a mass reduction of 65% of the tracker, which utilize the same antenna and substrate as the original, the mass of the tracker can still be seen as highly optimized.

Even though a large amount of time were focused on the design of new PCB's, it turned out that the enclosure were the part, for which the largest reduction could be performed. However the focus on the PCB wasn't a waste, since the enclosure couldn't be reduced as much, if the external modules wasn't moved to the main PCB. By moving the SAW filter and power converter onto the main PCB, the tracker was able to become flatter and therefore easier to fit into a smaller box.
The redesign of the PCB also opened up for the use of flex print PCB's, which can evolve into an even lighter tracker. This is new territory for DTU Space and has therefore not being looked into before. In this project, the use of flex print was tried and the difficulties, like the bending, was discovered. Future projects can therefore use this knowledge in the future and probably avoid these mistakes.

8 Further Work

Due to the mistake of the footprints, a functioning tracker has yet to be created. On top of this is there also a lot of other things that needs to be done, in order to create the best possible tracker. Some future work for future projects are listed below.

- The PCB's needs to be reordered with the corrected footprints.
- The trackers needs to be tested. This could be according to the proposed testing.
- If one of Design 1 or 2 functions properly, with the ceramic antenna, and the flex print design also functions acceptable, a new design could be created. This Would be a design which both utilizes the flex print and the ceramic antenna.
- The code can be power optimized, which can lead to a even smaller battery solution.
- Optimization of user friendliness of ground station tracker. When tracking the stratospheric balloons, the ground station follows the balloon in a car. However the data balloon comes as data, which then need to be filled into Google Maps, in order to follow the balloon. A script or program could be created, which directly shows the balloons location on a map, which would make tracking way easier.

References

[1] Cheng, David K., ”Field and Wave Electromagnetics”, 2. edition, Pearson 2014.

[2] Hampley, Alan R., ”Electrical Engineering Principles and Applications”, 6. edition, Pearson 2014.

[3] ATmel, ”ATmega 328P datasheet” https://4donline.ihs.com/images/VipMasterIC/IC/ATML/ATML-S-A0002642585/ATML-S-A0002642585-1.pdf?hkey=6D3A4C79FDBF58556ACFDE234799DDF0, November 2016.

[4] Semtech, ”SX1272/73 datasheet”, https://semtech.my.salesforce.com/sfc/p/#E0000000JelG/a/440000001NCE/v_VBhk1IolDgxwwnOpcS_vTFxPfSEPQbuneK3mWsXlU, January 2019.

[5] Texas Instruments, ”LMZ10503 power module datasheet”, http://www.ti.com/lit/ds/symlink/lmz10503.pdf, July 2016.

[6] EPCOS, ”869 SAW filter datasheet”, http://www.mouser.com/ds/2/136/B3715-84099.pdf, February 2008.

[7] C.I.F, ”Cobber-clad laminate datasheet”, http://www.farnell.com/datasheets/44643.pdf, October 2006.

[8] Panasonic, ”Felios R-F775/R-F786W datasheet”, https://docs.oshpark.com/resources/flex-substrate-Panasonic-Felios-F775.pdf, December 2013.

[9] Johanson Technology, ”868 MHz antenna datasheet”, http://www.farnell.com/datasheets/2280483.pdf, July 2016.

[10] Torex Semiconductor, ”XCL205/XCL206/XCL207 datasheet”, http://www.farnell.com/datasheets/2012857.pdf, January 2013.

[11] Energizer, ”Energizer L92 Ultimate lithium datasheet”, http://www.farnell.com/datasheets/1916377.pdf, Date N/A.