ECOLE POLYTECHNIQUE FÉDÉRALE DE LAUSANNE

SEMESTER PROJECT

EPFL ROCKET TEAM

Hermes II Avionics Redesign

Charlotte HEIBIG

Under the supervision of Alexandre Schmid and Iacopo Sprenger.

Spring 2022

Contents

List of Figures

List of Tables

Symbols and abbreviations

ACI	Atelier des Circuits Imprimés
ASSOC	Association
BOM	Bill of Materials
CM4	Cortex-M4
COTS	Commercial Off The Shelf
EPFL	Ecole Polytechnique Fédérale de Lausanne
ERT	EPFL Rocket Team
GPS	Global Positioning System
I2C	Inter-Integrated Circuit
IC	Integrated Circuit
IMU	Inertial Measurement Unit
MCU	Microcontroller Unit
PCB	Printed Circuit board
RF	Radio Frequency
RSSI	Received Signal Strength Indicator
RTC	Real Time Clock
SMD	Surface Mount Device
SPI	Serial Peripheral Interface
THT	Through-Hole Technology
USB	Universal Serial Bus
UART	Universal Asynchronous Receiver Transmitter

1 Introduction

1.1 Purpose and motivation

The EPFL Rocket Team is an association bringing together students whose goal is to build and launch rockets with various goals. Amongst these projects is the design and launch of the supersonic rocket Wildhorn at the Euroc competition in Portugal in October 2022, which aims for teams of students to develop, test and launch solid, hybrid and liquid rockets at various altitudes. Although there is a lot of literature about supersonic flights, the team lacks experience in this domain. Because some of the effects due to supersonic velocities are hard to model, the idea was born of building a test bench for Wildhorn to acquire data and test some new concepts. Building a model version of Wildhorn was considered the cheapest and safest option to test different critical aspects of supersonic flights. In 2020, Hermes I, the first iteration of this test bench was designed and flew for the first time in the subsonic realm in June 2021, and then two more times between June and October. Sadly, it never reached a supersonic velocity, as it exploded during its last supersonic launch in October.

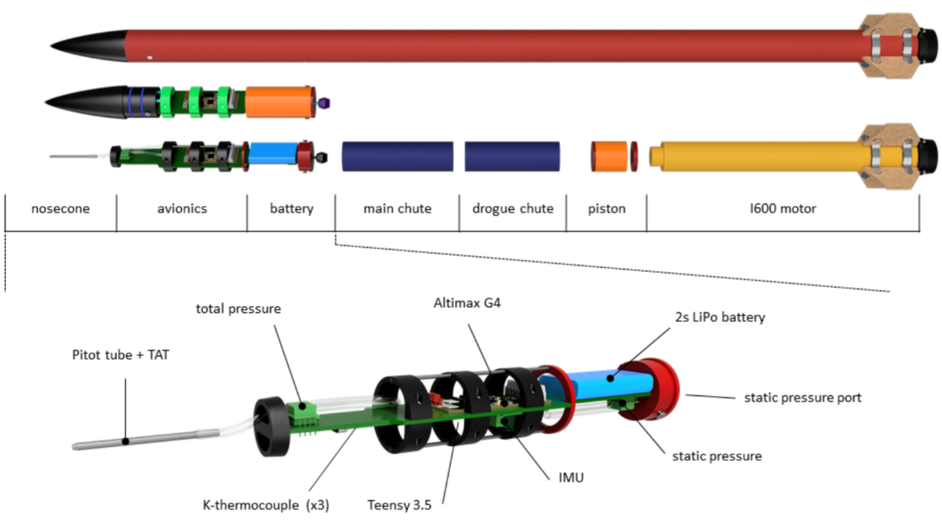


Figure 1: Hermes I structure overview and avionics bay

Hermes II, a second iteration of the rocket, was thought of and started taking shape a few weeks after Hermes I's last launch. Different elements were modified in this second version, including the avionics, which was supposed to be redesigned to be smaller, more efficient and to include more capabilities and sensors. The size of Hermes II is the one of Wildhorn, but with a 3:8 scale (38 mm is the diameter of the motor tube, and Wildhorn has a 100 mm diameter). It can go up to 2727 m with an i600 motor, at a speed of mach 1.55, and can support an acceleration of over 60g.

The goal of this semester project was to redesign the electronics of Hermes II, using as a basis the work of Joshua Cayetano-Emond and Bastian Winzer who created the first Hermes board.

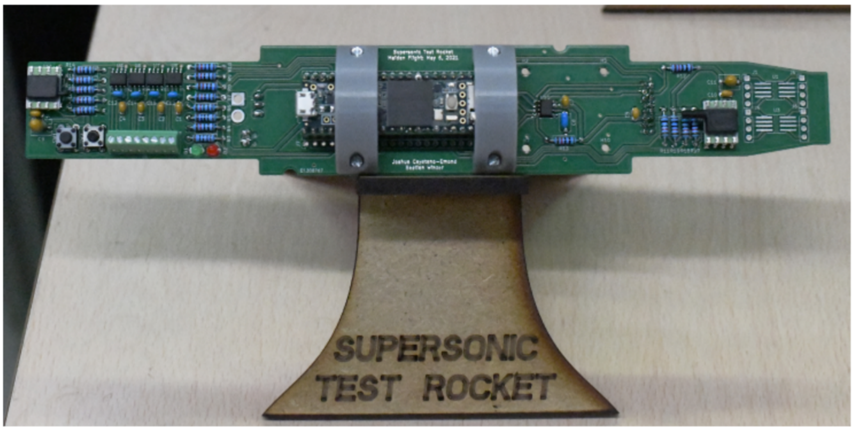


Figure 2: Board of the first Hermes rocket

1.2 Requirements

Requirement	**Explanation**
The main PCB shall be shrinked to approximately half of it's previous size	Leave some space for a potential payload
The electronics shall be separated in different parts/boards	Depending on the function of the sensors / IC s
The PCB shall include a GPS	Retrieve rocket after touchdown
The PCB shall have basic telemetric capabilities	Transmit data during flight in case of an explosion and retrieve the rocket after touchdown
The electronics shall include Wildhorn's sensor board	Test the sensor board at a supersonic speed
Include a load cell and a PCB to interface it with the main board	Measure the thrust of the motor
The data acquisition shall be fast enough	To acquire the most data during the supersonic regime which only lasts a few seconds
The sensors shall be redundant	In case of failure

Table 1: Hard requirements

Requirement	Explanation
The PCB should have a way to be powered without having to open the rocket	Reduce the time spend setting up the rocket on the launch pad
The PCB should include better battery management	Optimize the resources and adapt them to the new components
The PCB should include a battery protection circuit and a voltage indicator	Facilitate debugging
The entire electronic structure should be connected using a single wire harness	Facilitate the avionics setup
The PCB should include an RTC	Synchronize all the received data
The PCB should include improved thermocouple connectors	Facilitate the integration of the thermocouples

Table 2: Soft requirements

2 Method

2.1 Structure

To best accommodate the needs of the project, it was decided to make a main board and three auxiliary ones, which would be put in strategic parts of the rocket. The sensor board for Wildhorn should also be included, as well as a few other COTS components.
One sensor board to measure the total pressure and the temperature will be put in the nosecone, along with a pitot tube. Another similar board to measure the static pressure and the temperature will be put in the middle the rocket, and lastly a board to amplify the load cell signal will fit in a small space above the motor. The shape of the boards plays a special role in the project as it determines the amount of space available to place the ICs and to route the boards. Many iterations were needed to determine the final dimensions of the boards, which is constrained by the structure of the rocket.

The first sensor board (Figure 3) is a 40 mm x 24 mm board. It takes the shape of the nosecone, and is designed to slide in rails placed at both sides, so component placement is not allowed in this zone.

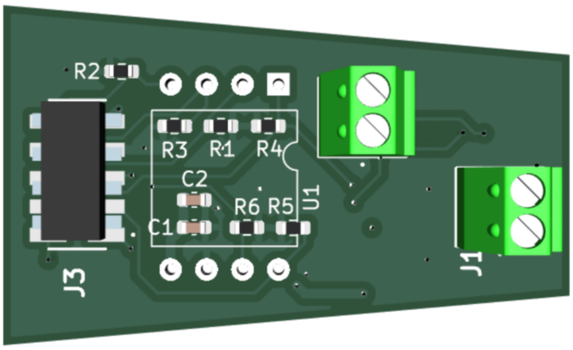


Figure 3: 3D render of the nosecone board

The second sensor board (Figure 4) is a 38 mm diameter circle (the diameter of the rocket) with two mounting holes to be screwed to the structure, and some enclosures for the cables that are required to connect the load cell and the pyros to the avionics bay.

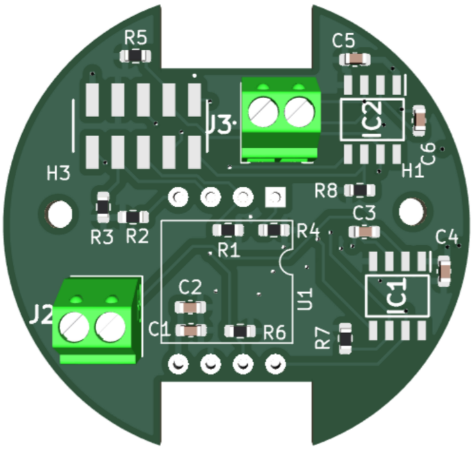


Figure 4: 3D render of the static port board

The load cell board (Figure 5) is the smallest board of all. It fits in a 19 mm diameter enclosure above the motor and also has two mounting holes to be screwed to the structure.

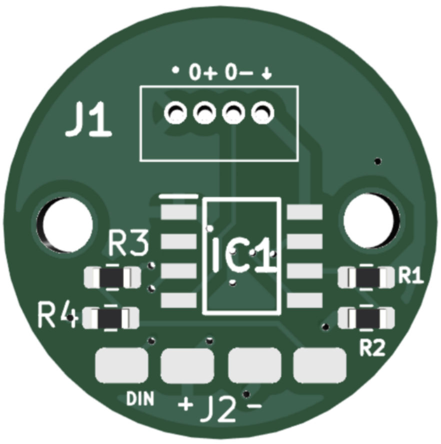


Figure 5: 3D render of the load cell board

The main board had to be as small as possible to make some space for a potential payload in the nosecone area. After trying to make it fit completely in the nosecone and coupler, it turned out that it was not possible, so it was pushed towards the back of the rocket, with the front of the board taking the shape of the nosecone and the back being rectangular. The final dimensions were 141.85 mm x

32.25 mm compared to 239 mm x 39 mm for the Hermes I board. Four mounting holes were added to attach the board to the avionics bay. Surrounding the board are the battery and board extensions which consist of an IMU, Altimax and the Wildhorn sensor board. Many iterations were needed to determine the placement of each component in the avionics bay, as we were limited by the height and width of the rocket. The space limitations strongly influenced the placement and choice of each component on the board.

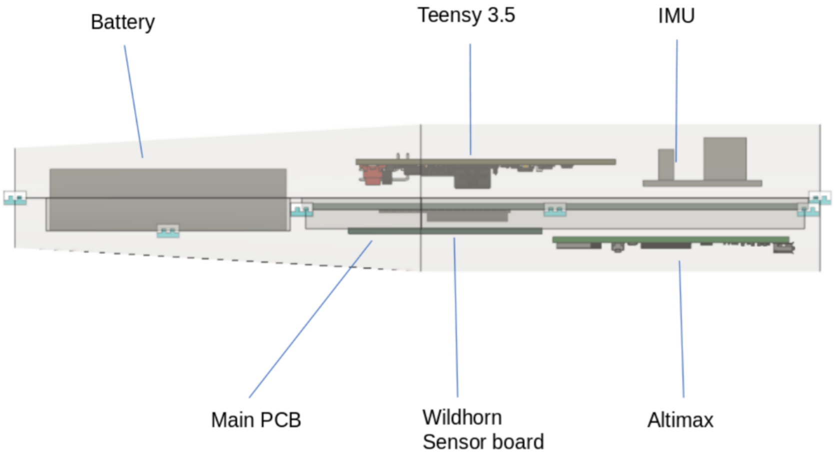


Figure 6: Cad of the avionics bay

The entire electronic structure is connected with a wire harness.
Two cables are also connected to the main PCB to turn on the pyros. The final result is the following :

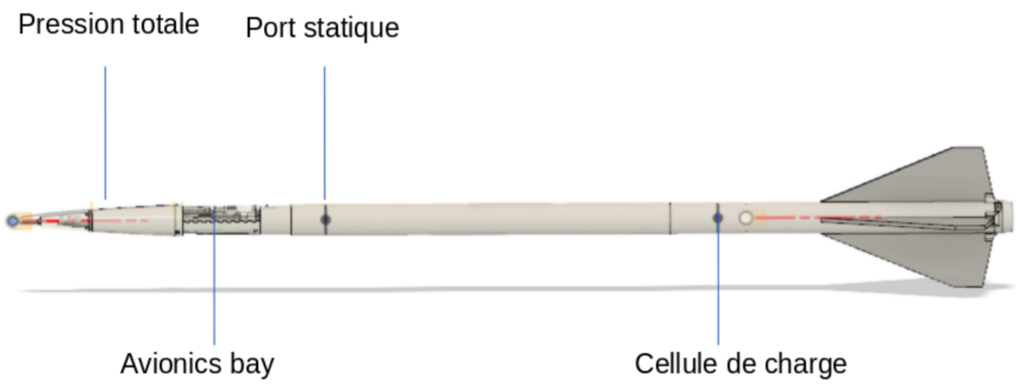


Figure 7: Overview of the Hermes II electronics

2.2 Choice of components

2.2.1 Microcontroller

The choice of microcontroller was motivated by the need of having a relatively simple to program device with multiple I/Os and relatively low power consumption. The decision was made to use the Teensy 3.5, which had already been Hermes I MCU. It was preferred to the Teensy 4.1's cortex M7 who has a much higher power consumption, or to the Teensy 4.0, who has much less I/Os. The Teensy 3.5 has 64 digital I/Os and 27 analog I/Os (mostly all 5V tolerant). As said above, the Teensy is easily programmed in the visual studio IDE in Arduino using an USB port. It also has 6 serial ports, as well as 3 SPI and 3 I2C ports. The first SPI port and serial ports 1 and 2 feature a FIFO for higher speed, which will be useful for our application as our data acquisition period is short. The Teensy's 3.5 main memory storage is based on an SD card. One disadvantage of choosing this microcontroller is that it is not in stock anymore.

2.2.2 Power supply and RTC

The choice of the power supply components was mainly motivated by their size, stock status, and their ease of use. The PCB is supplied by a 7.4 V 450 mAH lipo battery, which we had already used for Hermes I. The choice of this battery was motivated by our power budget, which gives us a total of 2h30 of operating time. The solution of upgrading to a battery with a bigger capacity was rejected because of weight and dimension constraints.

	T nominale (V)	I typical (mA)	power mWh	
Teensy 3.5	5	50	250	
Altimax G4	7,4	30	222	
Xbee SX	3,3	100	330	
GPS ublox	3,3	21	69,3	
Pressure sensor static	3,3	3,1	10,2	
Pressure sensor total	3,3	3,1	10,2	
IMU	3,3	50	165	
accelerometer	3,3	3	9,9	
thermocouple x4	3,3	4	13,2	
Traco	7,4	7	51,8	
E match (5s) x 3	7,4	1000	30	
load cell	3,3	3	9,9	
			1171,5	
pcb contigency	15,00 %			
total for 1h run time			1347.00	-> 182 mAh

Table 3: Power budget

We chose, considering the lack of space on the PCB, to solder the two battery wires directly on pads on the PCB. We used an XT30 connector at the other extremity. The Teensy also has a real time clock feature, which can be enabled by connecting a 3V coin cell to VBAT. The coin cell was to be as small as possible, however, the size of battery holders limited our choice to a 1205 battery. The chemical composition of the battery was imposed by the 3V voltage, as lithium has a nominal voltage of 3V. We chose a 3030 battery holder. The battery protection diode was chosen to be a Schottky diode because of its low dropout voltage, and the Teensy's analog pin protection diode was chosen to be a Zener diode for voltage limitation. The DC converters are tracopowers, chosen for their robustness and reliability. They are also easy to use, compared to power management ICs. We used a TSR2433 for the 5 to 3V3 conversion, and a TSR-1-2450 for the 7.4 to 5V, 1A conversion. The choice of the reed switch was limited, as they are usually not used for the application that we selected them for. They were bigger than regular switches, and could not withstand vibrations (max. 30G). We nevertheless chose to use the MISM-7R-10-15 magnetic reed switch, with the idea that even if broken at takeoff, we only needed to use it once per flight (which is obviously not an optimal solution). We chose to put it in the direction were it would bare the less mechanical constraints at takeoff (vertical to the ground), which is when the rocket has the highest acceleration. We chose to use a thermally resettable fuse, as it seemed like the easiest and most reliable option, as we do not need to unsolder/resolder it in case of a problem. Finally, the Altimax power supply connector was chosen to be a terminal block, so that we could take out the wires easily from the main PCB.

2.2.3 Pressure sensors

The choice of pressure sensors was limited by the current shortage of ICs. Hermes I used two different pressure sensors, but their combined use lead to false data acquisition or no data at all, so it was decided to replace the seemingly faulty sensor. However, it was impossible to find an absolute numerical pressure sensor with the same granularity and range. The second best choice was to use the RSCDJNI060PASE3 that had already been used for the total port for the two pressure PCBs. It still has a decent range of 4 bars. It has a 24 bits resolution and a speed of 2kHz.

2.2.4 Thermocouples, connectors and converters

The chosen thermocouples are K type thermocouples. This means that they are composed of chromel (+) and alumel (-). They are cheap, easy to find and quite reliable. We opted for thermocouples instead of a thermistor or an RTD because of their short response time. They can detect temperatures ranging from -200 to 1280 degrees Celcius. The thermocouples to digital converters are the MAX31855KASA+, which were the ones already in use for Hermes I. They have an ADC 14 bits and a frequency of 10Hz. The thermocouples cannot be soldered directly on the PCB, so a solution like crimping or screwing the wires had to be used. Hermes I used terminal blocks to screw the thermocouples on the PCB, but this solution was not easy to use because of the thin diameter of the wires. We tried looking for a simpler solution, but taking bigger wires meant a slower response time, and bigger connectors were simply too large to fit on the PCB. Finally, it was decided to keep the terminal blocks.

2.2.5 Accelerometer and IMU

We decided to add two accelerometers in the avionics bay. The first one, a one axis altimeter (AIS1120SXTR) is used only at takeoff, because of its saturation at 120 g of acceleration, compared to the second IMU, the ADIS16470. Indeed, this IMU is much more precise and reliable. It is a six axis device with a data acquisition speed of 2000 Hz. However, it saturates at 40g so will not be operational at takeoff, where the acceleration is much higher. Because it is a BGA component, it was bought directly soldered to a breakout board, in order for it to be reusable. Indeed, it is otherwise difficult to solder and unsolder such components.

2.2.6 RF module and antenna

The chosen radio is an XBEE XB8X DMUS 001. It is quite large but much easier to program than a smaller LoRa device, which was also a possible choice at first. The choice of this particular Xbee was motivated by the fact that it could transmit over a very long range (13 dBm of transmit power) which turns out to be much more than needed. Accordingly, it has a high power consumption. Its receiver sensitivity is 106 dBm at 80 kbps, which is the data rate that we chose. It emits in the 868-870 MHz range, and we will we using it at a 868 MHz frequency. We chose the option of having an integrated u.Fl connector, which is more reliable than the two other options of having an integrated chip antenna, or soldering directly an antenna connector on an RF pad. The chosen antenna is a patch 206764-0100 omnidirectional antenna. It has a linear polarization and a maximal gain of 1.2 dBi. The chosen ground station antenna is the helicoidal antenna from Bella Lui II, as well as the BL2 software, which would have been suited for our use. We calculated a link budget in order to ensure that communication is possible, or to know what the maximum range of operation with the chosen components is. The rocket is placed at a distance of 3 km maximum from the earth, and the ground station is at a 600 m of distance from the launchpad.

Element	Losses/ Gain
Receiver sensitivity	-106 dBm
Transmitted power	13 dBm
Worst antenna gain for the rocket	1.2 dBi
Gain or the ground station	15 dBi
Polarisation losses	-3 dB
Pointing mismatch	-10 dB
Losses in the cables etc.	-10 dB
Path losses	-100 dB
TOTAL	12.2 dBm

Table 4: Link budget

$P_{rx} = P_{tx} + G_t + Gr + L_{path} + L_{pol} + L_{pointing} + L_{mismatch}$

• P_{rx} is the received power, which should be above the receiver sensitivity. • P_{tx} is the transmitted power
• G_{tGt} the gain of the transmitting antenna
• G_{rGr} the gain of the receiving antenna
• $L_{path} = 20 \cdot Log\left(\frac{c}{4\pi D f}\right)$ the path losses, D being the distance of the transmission, f the operating frequency
• L_{pol} the losses due to polarization mismatch (linear vs helicoidal antenna)
• $L_{pointing}$ the losses due to pointing mismatch
• $L_{mismatch}$ due to diverse mismatches (cables, reflections, etc)

The received power is the power recieved at the receiver, and should be above the receiver sensitivity, which is our case here, as we have a margin of 12.2 dBm.

2.2.7 GPS

The GPS is the Ublox MAX 7C 0. The choice of this GPS brand was strongly motivated by the fact that the ERT has a sponsoring agreement with Ublox. If the GPS is connected to the 3V coin cell, it can have a hot start of 1s thanks to its RTC. The GPS will be turned off during the flight due to its saturation at over 4g of acceleration, and will only be useful during the descent and for recovering the rocket. It can be used in continuous mode and power save mode, which is useful for our application, as we do not have much embedded power. The chosen GPS antenna is a 1575 MHz omnidirectional ANTX100P001B15163 patch antenna with a linear polarization and an average gain of -1.31 dBi. It is connected to the GPS via an external u.Fl connector, as the device did not have any integrated solution for the antenna.

2.2.8 Load cell amplifier and load cell

The chosen load cell is the FX292X 100A 0200 L load cell. It can measure a force up to 1000 Newtons, but due to this high value, only an analog mV output was possible for the choice of the load cell. Its output ranges from 0 to 100 mV. This is why we had to amplify the signal before its arrival at the main pcb, a meter higher. The load cell amplifier is the AD855ARZ instrumentation amplifier. It was chosen over a usual differential amplifier for its precision and reliability. It is a 1 channel amplifier with a 5V differential input and a programmable 2 stages gain of 70 to 1280. It also has 2 modes : a programming mode and a simulation mode. It can also work in initial mode. A sufficient speed was also a requirement, and it is met with its 11.5 MHz frequency (still above the teensy ADC speed of 24 MHz). The ideal solution would have been the HX711 load cell amplifier, but it was discarded due to lack of space on the pcb.

2.2.9 Connectors

The choice of connectors was fastidious, as every component required a specific type of connector. For the interfacing with the other custom made PCBq, we chose standard 2 mm pitch TE connectivity connectors (for example the 1-2842102-0 10 position connector along with its female counterpart). For all PCBs, we chose SMd connectors with two rows in order to minimize their length and to limit save some space (compared to THT connectors). For the IMU, the header was already imposed by the breakout board. It is a 16 positions 2 rows 2 mm pitch male connector. The Altimax I/Os could be used using a micromatch connector. We chose to use a different connector at the other end of the cable, in order to save space on the main board and in an attempt to normalize the connectors. Therefore, we chose the same TE connectivity connectors as the ones for the sensor boards. The connectors for the thermocouples had to be terminal blocks, and the Altimax G4 connector was also chosen to be a terminal block (both cases discussed above). We chose to solder THT headers for the Teensy instead of soldering it directly on the board, so that it could be easily removed in case of a device malfunction. The load cell connector that was chosen for our application was the one suggested by the datasheet of the load cell (TE 1734598). On the other side, the cables coming out of the load cell were crimped.

2.2.10 Altimax G4

The Altimax G4 is a standard component in nearly all ERT rockets. It was chosen for its robustness and numerous features. It is in our case mainly used for apogee detection to open the parachutes and to trigger the pyros. Moreover, it also includes an altimeter and a barometer, along with 4MB of flash memory, which can be accessed by USB.

2.2.11 Wildhorn

The Wildhorn sensor board is composed of 2 barometers (MS5611-01BA03), two 3 axis gyroscopes (A3G4250D) and two 3 axis accelerometers (H3LIS331DL) . These sensors were chosen by the team at the beginning of the year. There are two of each sensor for redundancy.

2.2.12 Debug leds, switches and other components

We chose to put a total of five debug leds on the PCB. The first two are simple leds that can be turned on an off by the software, for software debugging purposes. One led is used to indicate the battery voltage, and is connected to an analog input of the Teensy. The two remaining leds are used to indicate signal strength coming in and out of the Xbee. Two switches were integrated in the avionics. They are standard SMD tactile switches (RKC2SJF170SMTR LFS). We chose to use SMD components as much as possible because of the lack of space on the boards. For the capacitors and resistors, we opted for 0603 components. A piezo buzzer (SMI-1027-T-5V-R) was initially supposed to be integrated on the board REF. However, it was discovered that it needed too much current to function, which would have destroyed the Teensy, so it was decided not to solder it in the end. We chose standard UDC ribbon cables of different lengths in order to connect the different PCBs together.

The components were mostly ordered on mouser and digikey.

2.3 Main PCB: circuit and PCB design

As a compromise between signal integrity and cost, it was decided to make a 4 layers PCB. The two external layers are signal layers and the two inside layers are respectively a power and ground layer. At first, it was also decided to put two power planes (external layers) and two ground planes (in the middle). After discussion, it was deemed better to only put one power plane (3V3) and one ground plane in the middle layers, as power planes on the signal layers would only act as antennas (even more for fast signals) instead of shielding the signals as we had thought before. The ground plane ensures a good flow of the return current. For an optimal use, it must not be interrupted by any signals. The power plane was also used for the 5V traces. Both power and ground planes provide a good shielding of the different signal layers. We also used via stitching in order to reduce impedance between planes and to shield the signals, particularly between the GPS ground plane and the main ground plane.

The distance between the signal layers and the power plane is 0.11 mm. This is important as we will need this information to compute the required trace impedance for the GPS antenna connection. The width of the traces were determined as to not have a temperature increase of over 10 degrees Celcius. The signal traces have a width of 0.5 mm, and the power traces, which can withstand up to 1.5 Amperes, have a width of 1 mm. The chosen pcb manufacturer, pcbway, has design rules regarding minimum track width and minimum space between two tracks. This is due to their manufacturing capabilities which leads to a minimum resolution for copper etching. This gives us a minimum for the tracks and track spacing of 0.101 mm. The traces should also be kept as short and direct as possible, while avoiding right angles, in order to reduce cross talk, impedance and parasitic effects which could limit the system's performance. As we had only two digital signals, one having an extremely short path to travel (battery voltage indicator), and the other one being amplified beforehand (the load cell signal), we decided not to add a analog signal plane, nor to separate the analog and digital grounds. We have chosen a via diameter of 0.6mm and a via hole of 0.3 mm. Those dimensions were partly based on what pcb way could manufacture, and partly on the place that we had on the board. Indeed, for the manufacturing part, the minimum via diameter/hole dimensions are 0.35/0.2 mm, but it is highly recommended to go above these values, to be sure that the vias are drilled correctly. To gain some space, micro vias were not an option, as they greatly increase the pcb cost.

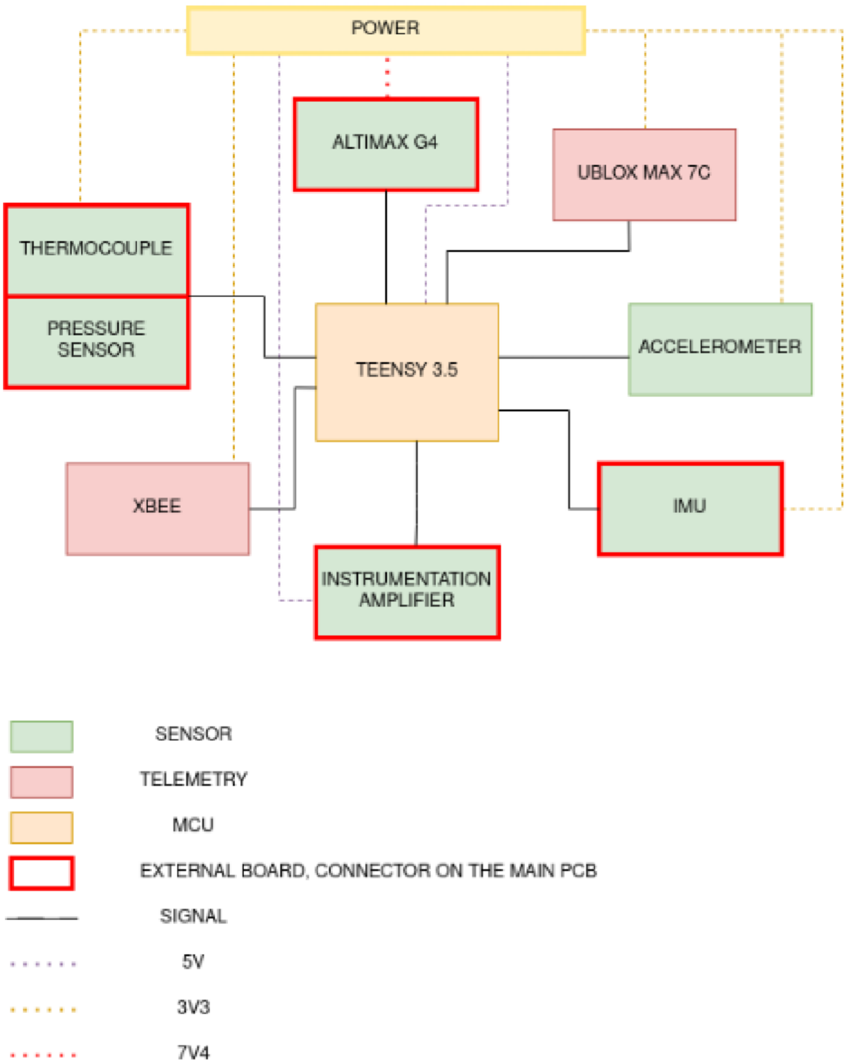


Figure 8: Block diagram of the main PCB

2.3.1 GPS

The GPS routing and placement was the most critical of the entire PCB. Indeed, the datasheet required the GPS to be placed far from any source of heat, RF emitting circuits, and digital circuits. Moreover, there were requirements concerning the power supply, which has to be stable, with wide power lines, and no source of resistance. The schematic below, taken from the datasheet, illustrates the correct device placement regarding the other pcb components, as well as the micro strip line and antenna placement.

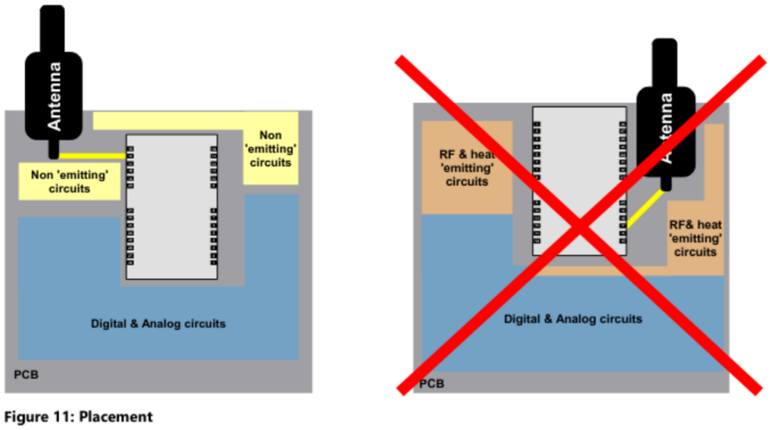


Figure 9: Correct placement of the GPS and antenna

The layout of the GPS has to follow precise rules in order for the communication to be optimal. The GPS should be placed on a ground plane, connected with as many vias as possible to the internal ground plane. The micro strip line should be shielded from any source of noise, again surrounded by vias and routed on a ground plane. No signals nor power lines should cross underneath the GPS module, because the ground plane only provides limited isolation. All of these elements are illustrated in the schematics below, both taken from the module's datasheet.

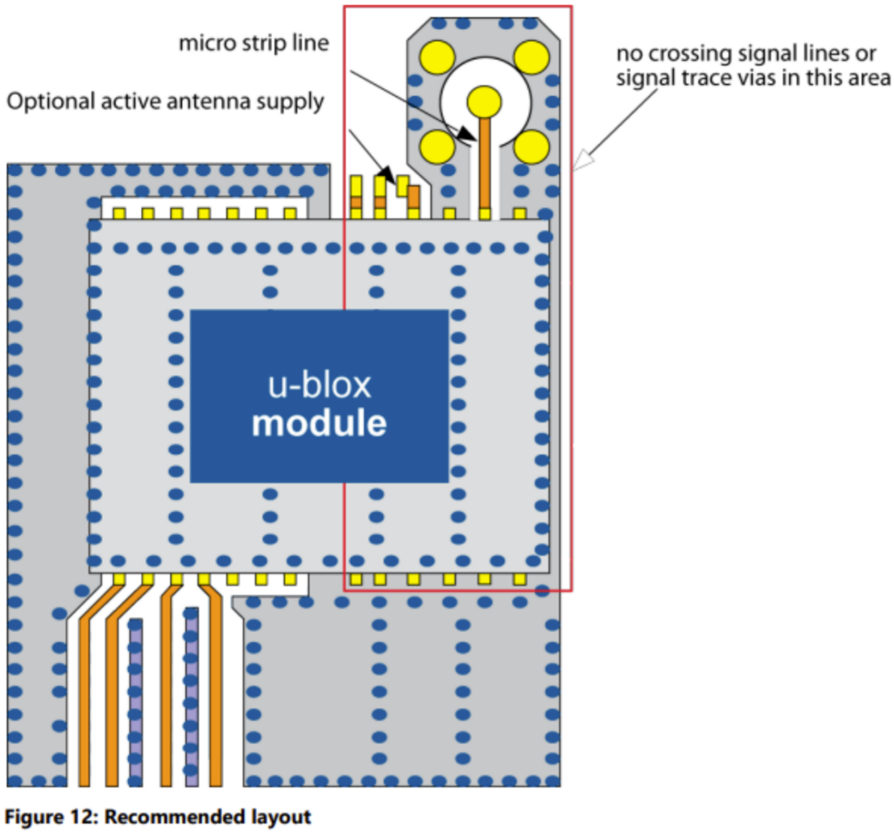


Figure 10: Recommended PCB layout

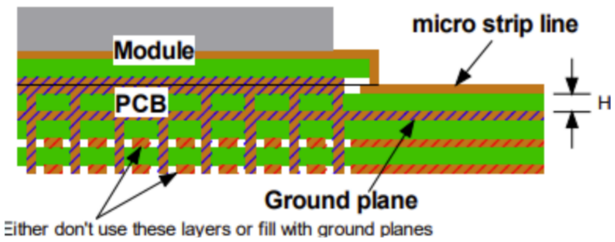


Figure 11: Module placement

The micro strip line has to have an impedance of 50 Ohm, and must be kept as short as possible. It must also be as directly as possible connected to the GPS, without any angles or turns. We used the KiCad trace width calculator to achieve the right micro strip impedance. The width W of the micro strip has to be chosen depending on the dielectric thickness H, the dielectric constant r of the dielectric material of the PCB and on the build-up of the PCB mechanical specifications as mentioned in the previous section. As seen on the picture below, this gives us a trace width of 0.190607 mm.

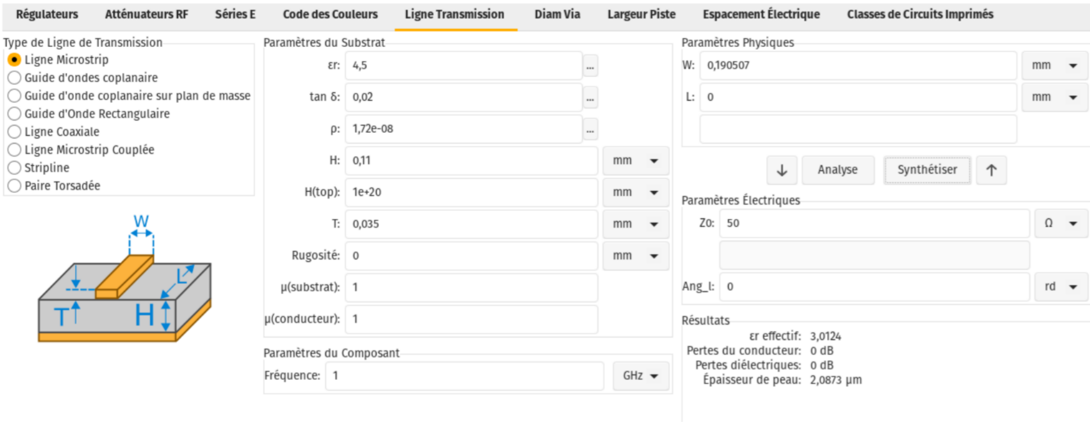


Figure 12: KiCad trace width calculator

The GPS is connected to the Teensy by UART. The timepulse output is connected to a digital input of the Teensy for debugging purposes. Finally, the antenna should be placed far away from any sources of metal of noise. It will be glued on the avionics bay, as far away from these elements as possible.

2.3.2 RF module

The main goal of the RF module is to transmit data during the flight in case of a crash or an explosion that could result in the destruction of the avionics. Its main purpose is also to relay the rocket's position at touchdown, so that it is easier to find the rocket. Again, in case of a crash, it would be possible to find pieces of the avionics, for example the main pcb, on which the GPS and radio are soldered. To facilitate the debugging when programming the RF module, we connected two leds to strategic I/Os of the module. We connected on led to the RSSI pin, and one led to the ASSOC pin. We chose to transmit data with the data rate of 80 kbps, which reduces the range, but was not a problem in our case as the XBEE was too powerful for our application. The radio antenna will be sticked inside of the avionics bay, as far as possible from an sources of metal. The radio is connected by UART with a speed of 921 kbps.

2.3.3 Connectors and accelerometer

The accelerometer is connected to the Teensy by SPI. It must be placed in the correct direction, as it is a one axis accelerometer. The connectors should be placed at strategic parts of the PCB, in order to minimize the length of the cable connections, and also to assure correct matching between the layout on the corresponding PCB.

2.3.4 Power management

In order to reduce the time spent on the launchpad, we needed a means to turn on the avionics without having to open the rocket. We therefore opted for a magnetic switch. The circuit was based on the following idea : a mosfet and the reed switch control the circuit connection to the ground. When we first power the pcb, the magnetic switch and the mosfet are both ‘open’, so the ground is not connected to the rest of the circuit. Therefore, the current cannot flow. When we approach a magnet near the switch, the ground is connected to the rest of the circuit for a short period of time, but long enough for the teensy to be powered on and the software to start running. The software sends the command to the teensy to apply a 3V3 voltage on the gate of the mosfet, so the connection to the ground is made on the side of the mosfet. The reed switch then plays no role in the control of the circuit connection to GND. To power off the circuit, the software must simply send the command to stop applying 3V3 on the gate of the mosfet.

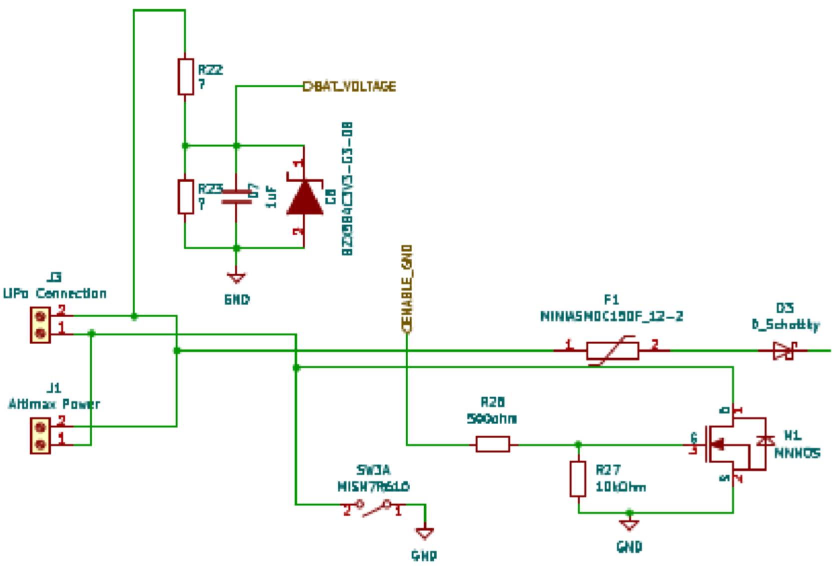


Figure 13: Means to turn on the avionics without opening the rocket

We also added a thermally resettable fuse and a schottky diode in order to protect the circuit and the battery respectively. The battery voltage indicator is composed of a led and an analog output that goes directly to an analog input of the teensy, before having been reduced by a voltage divider in order for it to not destroy the teensy (analog inputs being only 3V3 tolerant). A zener diode with a 3V3 reverse voltage is put in parallel to the analog indicator, in order to provide further protection for the teensy.

The two tracopowers are connected in series, the first one supplying a 5V voltage and 1 amperes, and the second one 3V3 and 1 ampere. The capacitors before and after the DC converters are used to stabilize and filter the power line.

Mostly all components are connected to the output of the 3V3 tracopower. The load cell amplifier and teensy are connected to the 5V. The altimax is connected directly to the 7.4V.
To be able to power the Teensy by battery and USB at the same time, we disconnected two pads VUSB and VIN and reconnected them with a diode.

2.4 Interfaces : circuit and PCB design

2.4.1 Sensor boards

The main PCB is connected to 3 sensor boards.

The first sensor board is in charge of measuring the total pressure. The total pressure is defined as the sum of the static pressure, the dynamic pressure (which is the kinetic energy of the fluid per unit volume), and the volumic density of the potential energy of gravity. The sensor is connected to the Teensy by SPI. It is linked to a pitot tube that we added in the tip of the nosecone.
The two thermocouples are glued to the walls of the nosecone, as to perform a temperature gradient. Thermocouples are based on the Seebeck effect. Two different materials are soldered together in a specific way (which we call the hot junction junction), and have different responses to different temperatures, which creates a voltage. The other sides of the wires are connected to the terminal blocks. We call this junction cold junction. Here is a schematic of the thermocouple setup.

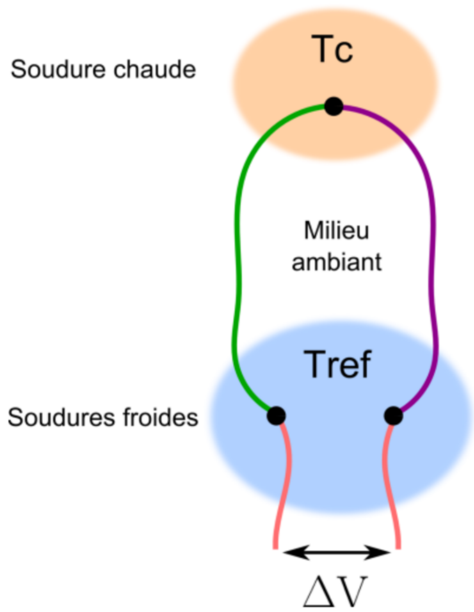


Figure 14: Thermocouple setup

The signal from the thermocouples are then sent to the thermocouple to digital converters, which is connected by SPI to the teensy.

The second sensor board is the board placed at the static pressure port. The static pressure is defined by, for a sensor in a fluid, the speed of the sensor relative to the fluid. A tube is also connected to the pressure sensor and to a hole in the rocket's fuselage, placed at a strategic part of the rocket (the simulations were made by the rocket's mechanical engineers). Two other thermocouples are glued to the side of the rocket.

These two boards perform different tasks, but their schematic is exactly the same, as they are composed of the same components.

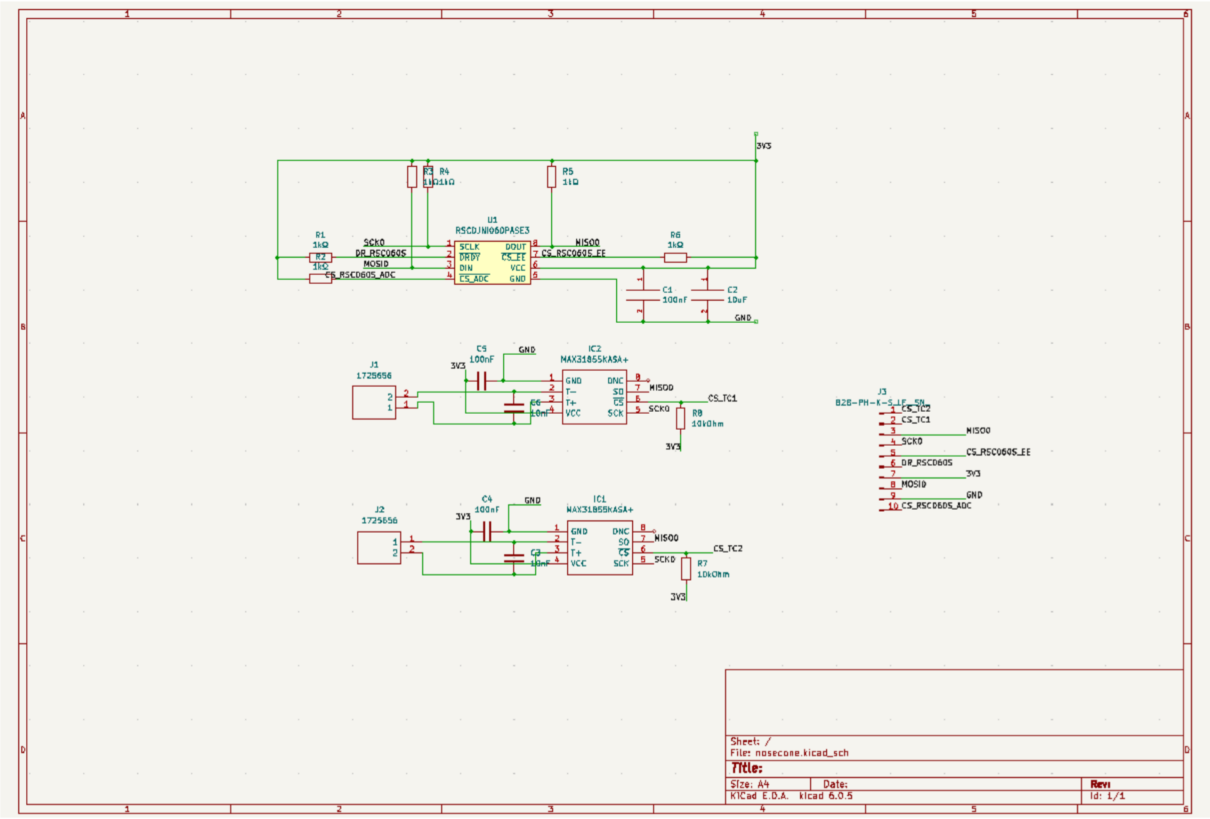


Figure 15: Schematic of the total and static port

The third sensor board consists of the Wildhorn sensor board. Some traces had to be rerouted in order for the header to be placed on only on side of the PCB. Otherwise, we wouldn't have had the space to connect the board to the main PCB. Their is a pair of each sensor. In each pair, a sensor is connected using I2C and the other SPI.

2.4.2 Load cell PCB

We measure directly the thrust of the motor with a load cell, which is basically a Wheatstone bridge. The four I/Os must be connected to a differential 5V and to two analog outputs.

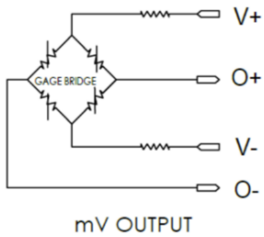


Figure 16: Schematic of the load cell interior

The signals were amplified using the instrumentation amplifier. However, the output range of the load cell being 100 mV and the minimum gain of the amplifier being 70, the maximum output voltage would have been 7V. The amplifier has a VCLAMP I/O so we can clamp the voltage to 5V as to not damage the teensy. The result would have been the clipping of the signal, and so a loss of information. To address this problem, we chose to put a voltage divider at the output of the amplifier, which is not the ideal solution, but the cheapest and the quickest. We can see the voltage divider on the schematic below.

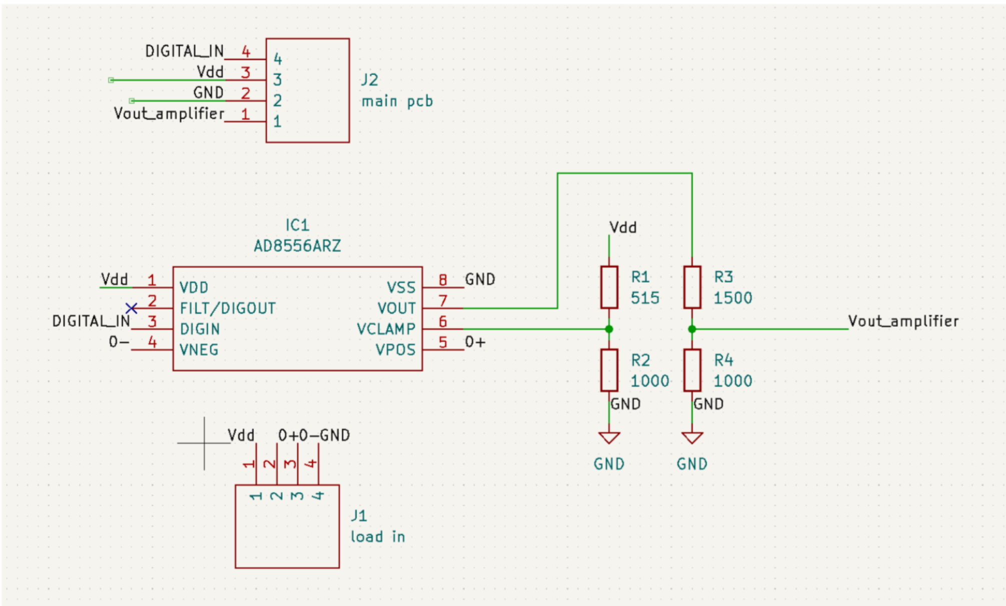


Figure 17: Load cell PCB schematic

2.4.3 Sensor board 2.0

Due to some issues, we had to manufacture a sensor board on a breakout board. We used the sensors that were initially planned for the PCB, and connected them with hookup wire. Down below are some picture of the results and the intermediate product.

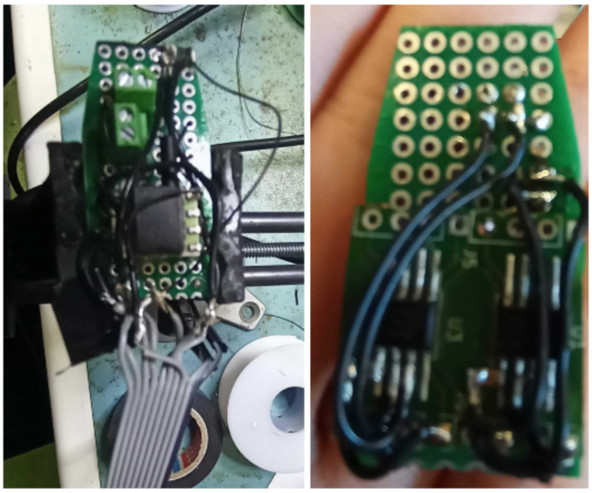


Figure 18: Sensor board 2.0

The PCB design rules (minimum via size and minimum track size) are the same as for the main PCB. All of the auxiliary are 2 layers PCBs.

2.5 Board manufacture

The board soldering was mostly done at the ACI at EPFL, using the pick and place machine and a vapor phase oven. For the Wildhorn sensor board, it was easier to use the micro placer, as the IC s were extremely small. To use these machines, we first need to apply solder paste on the pcb using the stencil provided by the manufacturer. We then start by placing all capacitors, resistors, and small SMD components, and we can then place the bigger ones. The THT components must be soldered by hand. However, as many components/boards hadn't arrived yet, we were not able to solder them at the ACI, and they had to be soldered by hand.

3 Results

We had some issues with testing the hardware, as the launches were all cancelled. We could however test some of the hardware in everyday situations. The PCB can be powered by USB and battery simultaneously without burning the USB port.

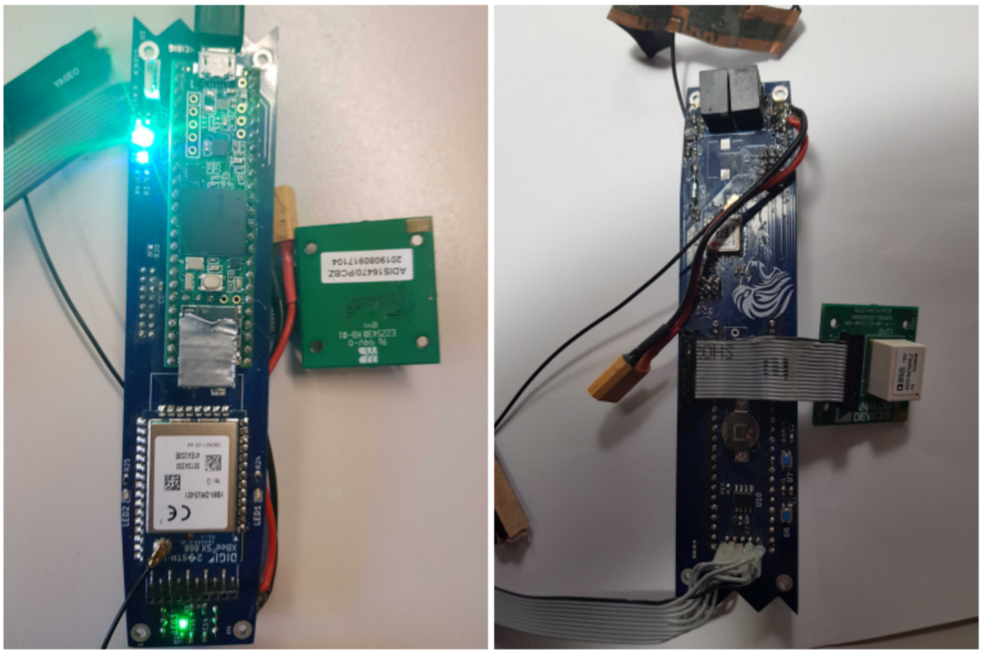


Figure 19: PCB powered by USB

The bright leds are those of the battery voltage indicator, the RSSI led, and the two debug leds.

The power circuit, however, does not work. This is probably due to the fact that another ground is connected somewhere else, which would allow the current to flow. This will be investigated further.

The accelerometer, IMU and altimax work correctly. The accelerometer still needs to be calibrated, or the data can be corrected by software after the flight.
The connectors were for the auxiliary circuit were wrongly oriented, which led to having to solder the cables on by one on the main PCB. After this procedure, however, the auxiliary PCBs seemed to work correctly, except for the burnt IC s, which gave us the error message that there were short circuits between nearly every pin of the IC.

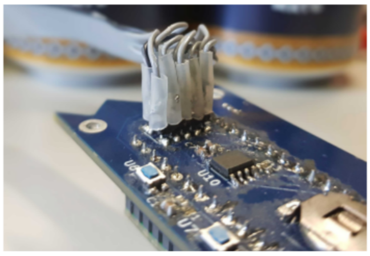


Figure 20: Cables soldered on the header

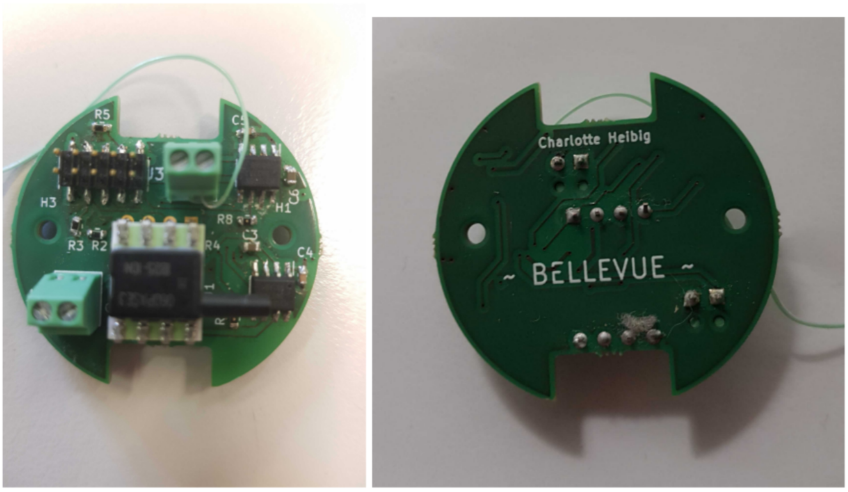


Figure 21: Static port PCB

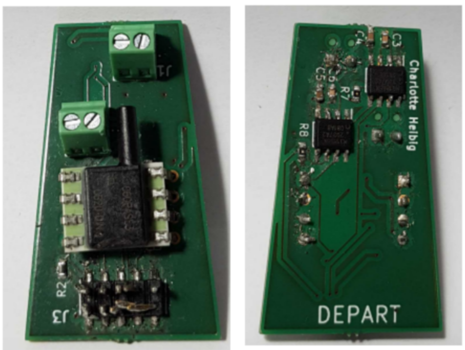


Figure 22: Total port PCB

The battery wire soldered directly onto the pad were unstable and disconnected themselves all the time, which led to us having to put epoxy on them. In the future, holes would be a better idea to host cables. The Wildhorn pcb was tested with a logic analyzer and the sensors respond correctly. However, as the code has not be written yet, we still have to test them with the software, which will be done during the summer.

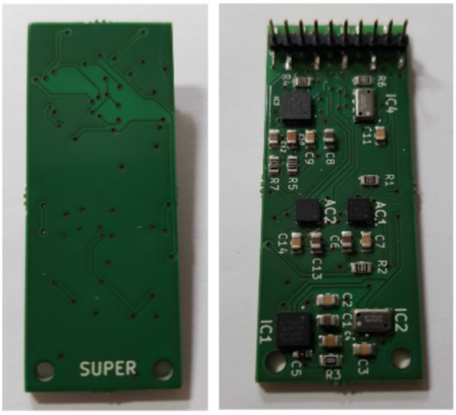


Figure 23: Wildhorn sensor board

After a few problems with the baud rate, the RF module was able to transmit and receive packets. The module was thoroughly tested by Michael in the scope of his semester project.

The load cell pcb software does not work at the moment, so the pcb will have to be tested later.

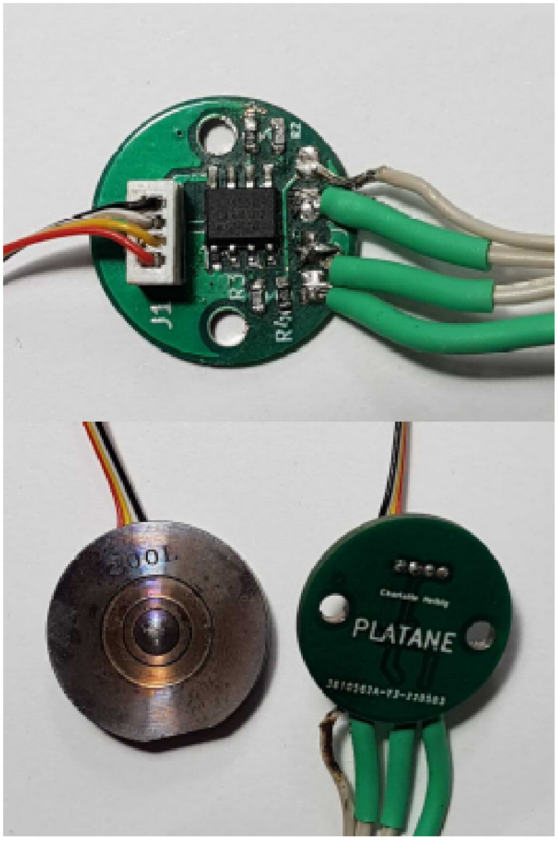


Figure 24: Load cell PCB and load cell

4 Discussion and conclusion

4.1 Launch and setbacks

There were many setbacks during the project. Certain components like the Teensy 3.5 and the thermocouples to digital converters were not in stock anymore, so we had to rely on the ERT stock. For the converters, we had to unsolder them, as the only ones that we had were on the previous Hermes board. This lead to two of them out of four not working. We will try to replace them by other IC s with a similar function and same footprint, as to not have to reroute and reorder a PCB.

A second problem that arose was the lack of time and the tight schedule. Indeed, the launch was scheduled for the 26 of march, and the project started about a month and a half before. This led to having only 3 weeks to do the PCBs, to then have the time to order them along with their components, solder them, and test the software. The main PCB was ordered in time, but the auxiliary ones were ordered a bit later, which lead to them arriving a day after the launch. This was also mainly due to the fact that the war in Ukraine led to long delays for their manufacture and shipment (even though they were produced in Europe).

This setback led us to manufacture a sensor board on a breadboard on the night before the launch (as shown in the pictures above). It was a very useful experience, in terms of technical skills gain. A requirement for the rocket to fly was that the GPS was operational, in order for us not to lose the rocket – and therefore the team's work and money – in case of a crash (or not). Sadly, the student responsible for software did not manage to make the GPS work, and the first launch was canceled. Other sensors were still not working at this point, as we had not had the time to debug the software. We had another launch opportunity in may, but the altitude limit was too low (600 m) and it requirement us to launch at a subsonic speed and therefore with a smaller motor. This would have requires us to change the entire structure of the motor retainer, and it was decided by the team that we would not proceed with the modifications, that were not enough rewarding in terms of work vs. gain. We nearly launched the avionics in another rocket's nosecone, but the battery burned on the launch site, which ended up saving the PCBs, as said rocket crashed an destroyed the other two PCBs that were inside.

We are now awaiting the next Cernier launch in October in order to fly Hermes II for the first time, at a supersonic speed.

4.2 Difficulties encountered during the project

The main difficulty encountered during the beginning of the project was to understand the requirements of Joshua, as well as his previous work, as I had never developed a PCB before. During the routing phase, the lack of space was the main issue, which made me sometimes overlook some elements in favor of making everything fit into the small boards. Further along in the semester and before the launch, the collaboration between all the members of the team turned out to be the biggest challenge, as all the subsystems had to come together, and strongly relied on each other to function.

4.3 Conclusion and future developments

Despite all the difficulties encountered during the semester, the project was extremely rewarding in terms of technical knowledge gain and from a personal point of view. It is not yet really finished, and will be further advanced during the summer, as we hope to be able to finally launch at Cernier in October 2022. We then hope in the future to reuse the avionics to test other aspects of supersonic flights. Indeed, the project Hermes III, exploring active drag control, was launched in March 2022 and will be using the boards developed for Hermes II, with some few improvements and additional features like a camera module and more thermocouples.

4.3.1 Acknowledgements

I would like to thank Joshua Cayetano-Emond for his precious advice during the time of this project, and for his work on the Hermes I electronics, which allowed me to have a strong basis to rely on. I would also like to thank Bastian Winzer for his work on the previous Hermes I avionics, and for his moral support during the entirety of the project. I am thankful to Iacopo Sprenger for his continuous support and advice during the semester. I would also like to thank Alexandre Schmid for his supervision and René Beuchat for taking the time to review my project. Finally, I thank the EPFL Rocket Team members who supported me throughout the year and without whom this project would not have been possible.

References

[1] Datasheet. Ad8555, zero drift, digitally programmable sensor signal amplifier. *Analog Devices*, 2015.

[2] Datasheet. Max31855 cold junction compensation thermocouple to digital converter. *Maxim Integrated*, 2015.

[3] Datasheet. Adis 16470, wide dynamic range, miniature mems imu. *Analog Devices*, 2019.

[4] Datasheet. Ais2120sx, mems automotive acceleration sensor: single/dual-axis with spi interface. *STMicroelectronics,*, 2020.

[5] Datasheet. Honeywell sensing trustability rsc series datasheet. *Honeywell*, 2020.

[6] Datasheet. Altimax g4 manual. *Rocketronicks*, 2021.

[7] Datasheet. Max-7 / neo-7 hardware integration manual, ubx-13003704. *Ublox*, 2021.

[8] Datasheet. Max7 datasheet ubx 13004068-2010070. *Ublox*, 2021.

[9] Teensy Documentation. https://www.pjrc.com/teensy/.

[10] KiCAD EDA. https://www.kicad.org/.

[11] Application Note. Tsr1-537631. *Traco Power*, 2020.

[12] Report. Fusée supersonique à échelle réduite pouvant être utilisée comme banc d'essai pour la validation de concepts et de technologies. *Bastian Winzer*, 2021.

[13] Report. System engineering for the design of a model sounding rocket. *Joshua Cayetano-Emond*, 2021.

A Appendix

Id	Référence	Boîtier	Quantité	Désignation
1	H4,H3,H1,H2	MountingHole_2.2mm_M2_Pad_TopBottom	4	Board_Hole
2	D1	LEDC1608X45N	1	White LED
3	C5,C14,C11,C3,C1,C2,C13,C10	C_0603_1608Metric	8	100nF
4	REF**	Charlotte	1	Charlotte
5	R3	R_0603_1608Metric	1	100Ω
6	IC2	XBP9XDMRS001	1	XB8X-DMUS-001
7	R20,R6,R4,R12,R11,R13,R1	R_0603_1608Metric	7	10kΩ
8	LED3,LED1,LED2	LEDC1608X80N	3	150060GS75000
9	R24,R25	R_0603_1608Metric	2	24
10	R14	R_0603_1608Metric	1	10K
11	R7,R8,R27	R_0603_1608Metric	3	10kOhm
12	D3	V8PAN50M3I	1	D_Schottky
13	R15,R16,R18,R10,R9,R2	R_0603_1608Metric	6	10k
14	J7	17345984	1	1734598-4
15	U3	HUSRSP10W50P200_2000X200X550P	1	1-2842102-0
16	J2	878980304	1	878980304
17	U4	Teensy35_36_exterior_pins_only	1	Teensy3.5
18	F1	MINISMDC050F2	1	MINIASMDC150F_12-2
19	R5	R_0603_1608Metric	1	24Ω
20	D2	LEDC1608X80N	1	Red LED
21	J3	pad2	1	LiPo Connection
22	U6,U7	RKC2SJF170SMTRLFS	2	switch
23	J4	CONUFL001-SMD	1	CONUFL001-SMD-T
24	IC4	TSR-0.5-2433	1	TSR_0.5-2433
25	C6	C_0603_1608Metric	1	1uF +/- 10% 10V
26	VR1	CONV_TSR1-2450	1	TSR1-2450
27	C9,C12,C4	C_0603_1608Metric	3	10uF
28	C8	C_0603_1608Metric	1	1uF
29	U10	SOIC127P600X175-8N	1	AIS1120SXTR
30	D7	SOD2513X110N	1	BZX584C3V3-G3-08
31	R26	R_0603_1608Metric	1	R
32	U9	PinHeader_2x08_P2.00mm_Vertical	1	ADIS16470
33	R22,R23	R_0603_1608Metric	2	?
34	IC1	MAX7C0	1	MAX-7C-0
35	J5	57202G5203LF	1	57202G5203LF
36	L3	SMI1027T5VR	1	SMI-1027-T-5V-R
37	J1	TerminalBlock_Phoenix_MPT-0,5-2-2.54_1x02_P2.54mm_Horizontal	1	Altimax Power
38	M1	SOT95P237X112-3N	1	MNMOS
39	SW3	MISM7R610	1	MISM7R610
40	U1,U2	57202G5205LF	2	57202G5205LF
41	U5	3030	1	3030
42	C7	C_0603_1608Metric	1	100nF +/- 10% 10V

Figure A.1: Main PCB BOM

Id	Référence	Boîtier	Quantité	Désignation
1	C6,C3	C_0603_1608Metric	2	10nF
2	IC2,IC1	SOIC127P600X175-8N	2	MAX31855KASA+
3	C5,C4,C1	C_0603_1608Metric	3	100nF
4	H1,H3	MountingHole_2.2mm_M2	2	MountingHole
5	R1,R3,R2,R4,R6,R5	R_0603_1608Metric	6	1kΩ
6	J1,J2	terminal block	2	terminal block
7	C2	C_0603_1608Metric	1	10uF
8	R8,R7	R_0603_1608Metric	2	10kOhm
9	J3	57202G5205LF	1	B2B-PH-K-S_LF__SN_
10	U1	DIP-8_W13.08mm	1	RSCDJNI060PASE3

Figure A.2: Static port PCB BOM

Id	Référence	Boîtier	Quantité	Désignation
1	U1	DIP-8_W13.08mm	1	RSCDJNI060PASE3
2	C1,C4,C5	C_0603_1608Metric	3	100nF
3	R6,R1,R5,R4,R2,R3	R_0603_1608Metric	6	1kΩ
4	J2,J1	1725656	2	1725656
5	J3	57202G5205LF	1	B2B-PH-K-S_LF__SN_
6	C2	C_0603_1608Metric	1	10uF
7	R8,R7	R_0603_1608Metric	2	10kOhm
8	C3,C6	C_0603_1608Metric	2	10nF
9	IC1,IC2	SOIC127P600X175-8N	2	MAX31855KASA+

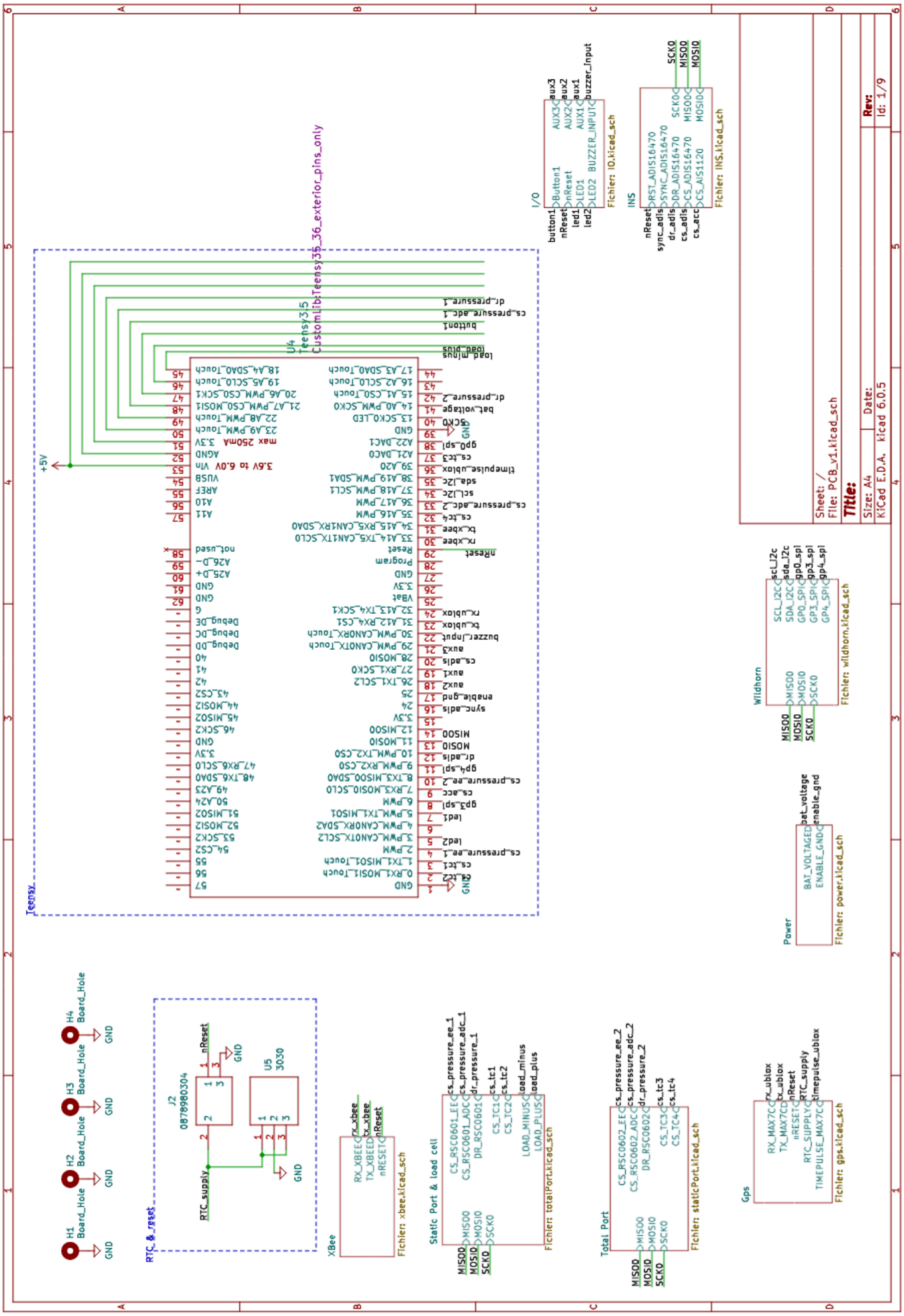
Figure A.3: Total port PCB BOM

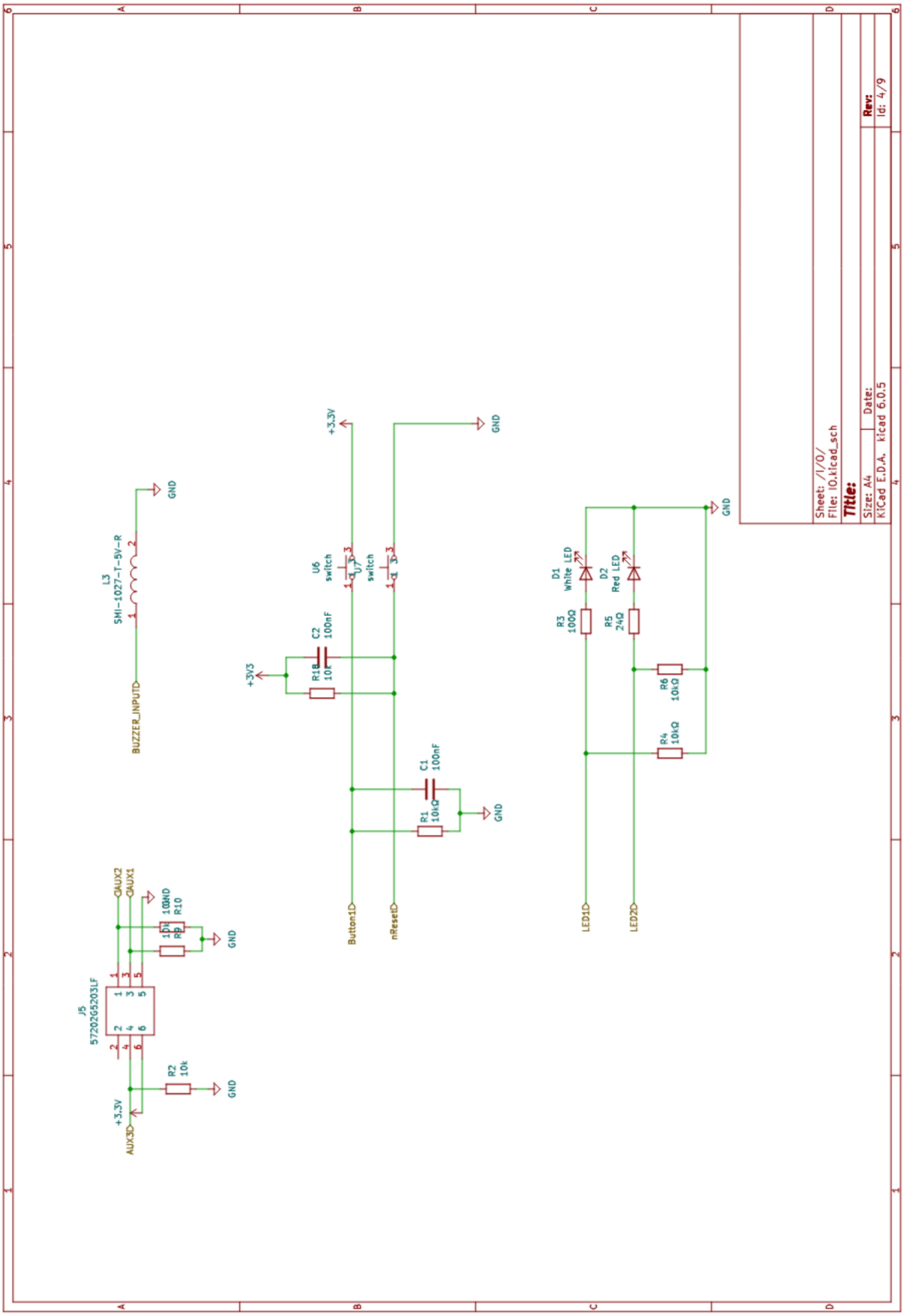
Id	Référence	Boîtier	Quantité	Désignation
1	R1,R2	R_0603_1608Metric	2	R
2	J1	17345984	1	load in
3	R4	R_0603_1608Metric	1	1000
4	IC1	SOIC127P600X175-8N	1	AD8556ARZ
5	J2	load to main	1	main pcb
6	R3	R_0603_1608Metric	1	1500

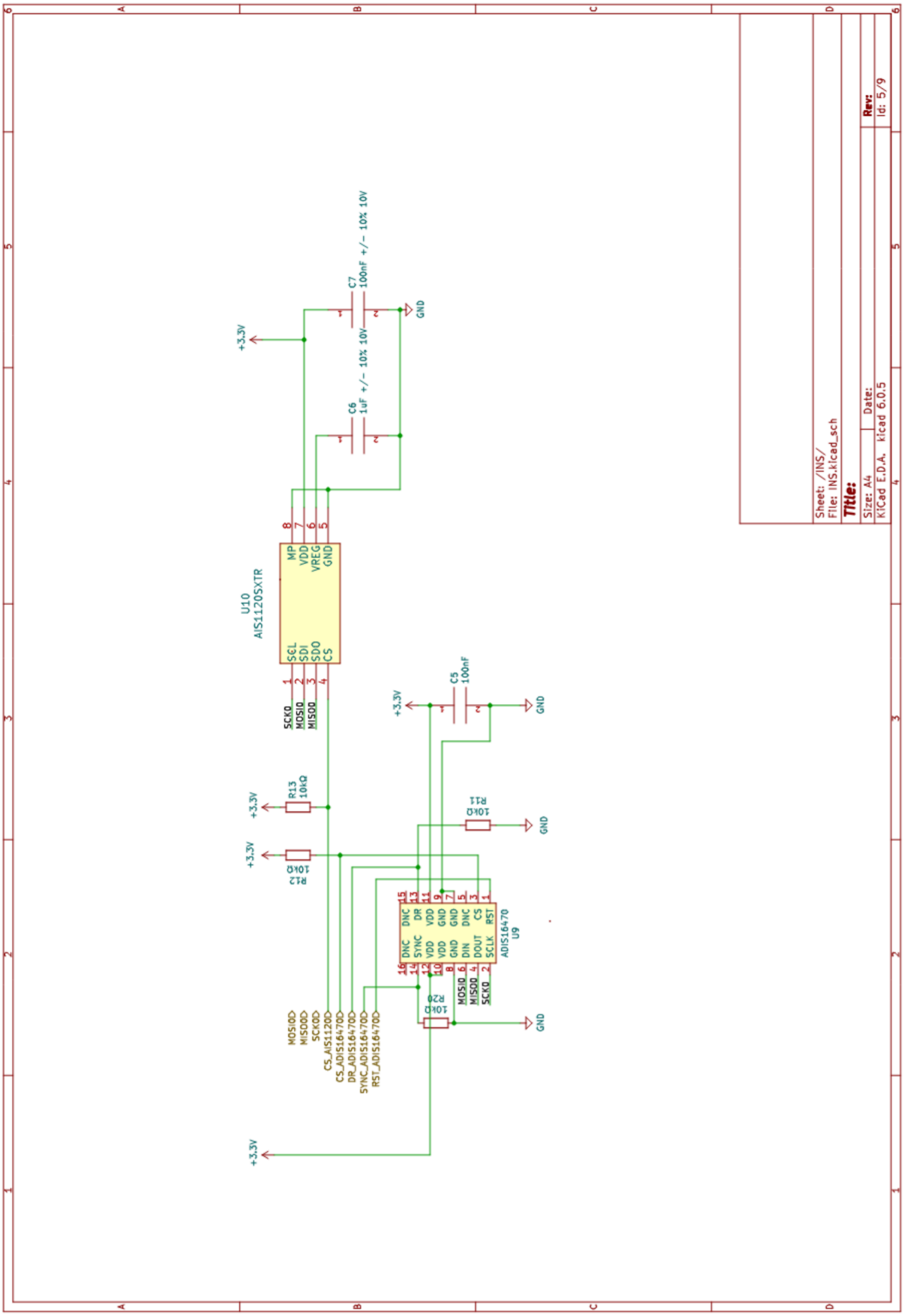
Figure A.4: Load cell PCB BOM

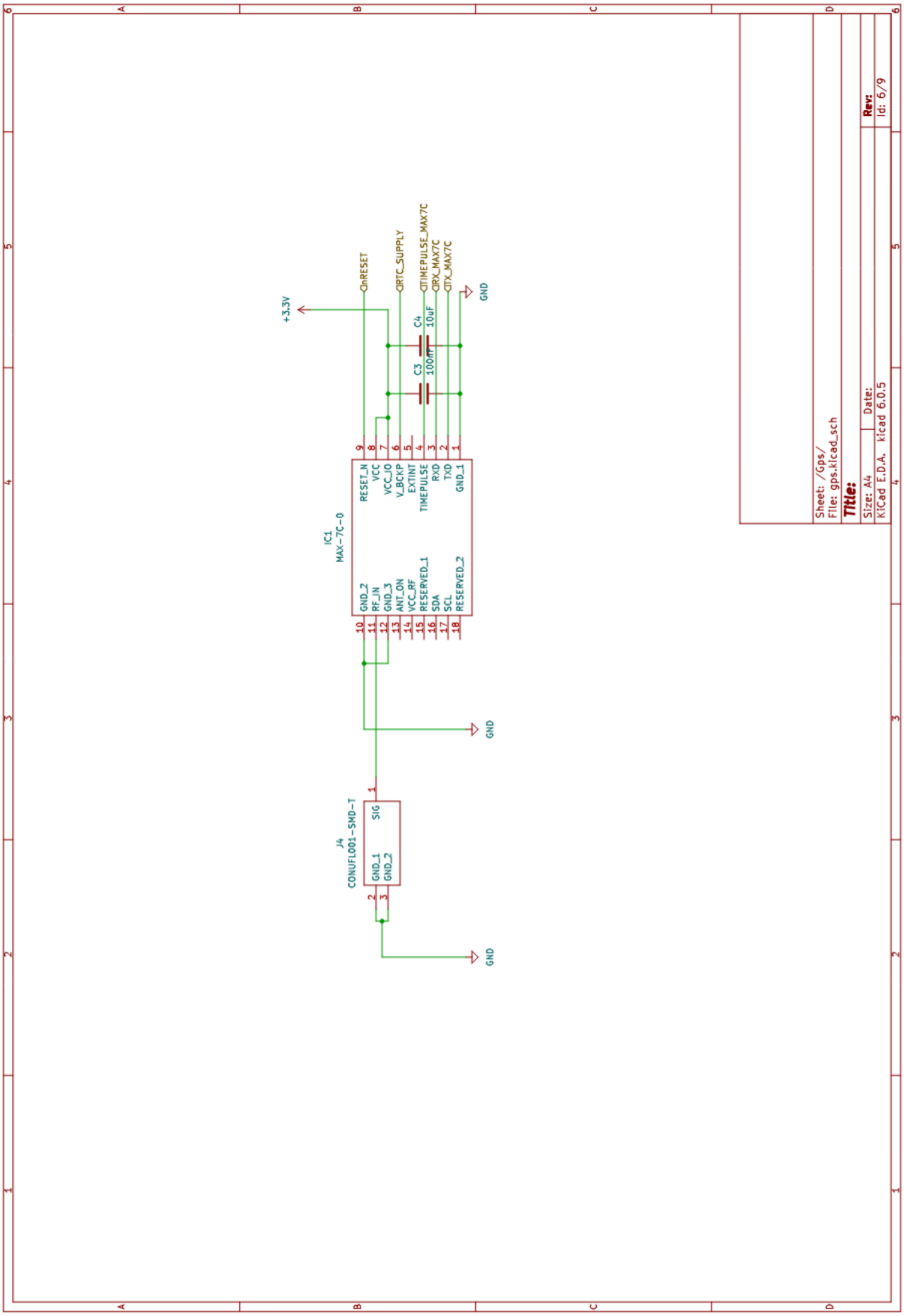
Id	Référence	Boîtier	Quantité	Désignation
1	AC1,AC2	3_Axis_Acc	2	H3LIS331DL
2	C1,C6,C8,C13	C_0805_2012Metric	4	10uF
3	C2,C3,C7,C9,C11,C14	C_0805_2012Metric	6	100nF
4	C4,C10	C_0805_2012Metric	2	470nF
5	C12	C_0805_2012Metric	1	10nF
6	IC1,IC3	3_Axis_Gyro_LGA-16L_4X4X1.1MM_	2	A3G4250DTR
7	IC2,IC4	Pressure_Sens	2	MS561101BA03-50
8	R1,R2,R3,R4,R5,R6,R7	R_0805_2012Metric	7	10kohm
9	J1	HUSRSP10W50P200_2000X200X550P	1	1-2842102-0
10	C5	C_0603_1608Metric	1	10nF

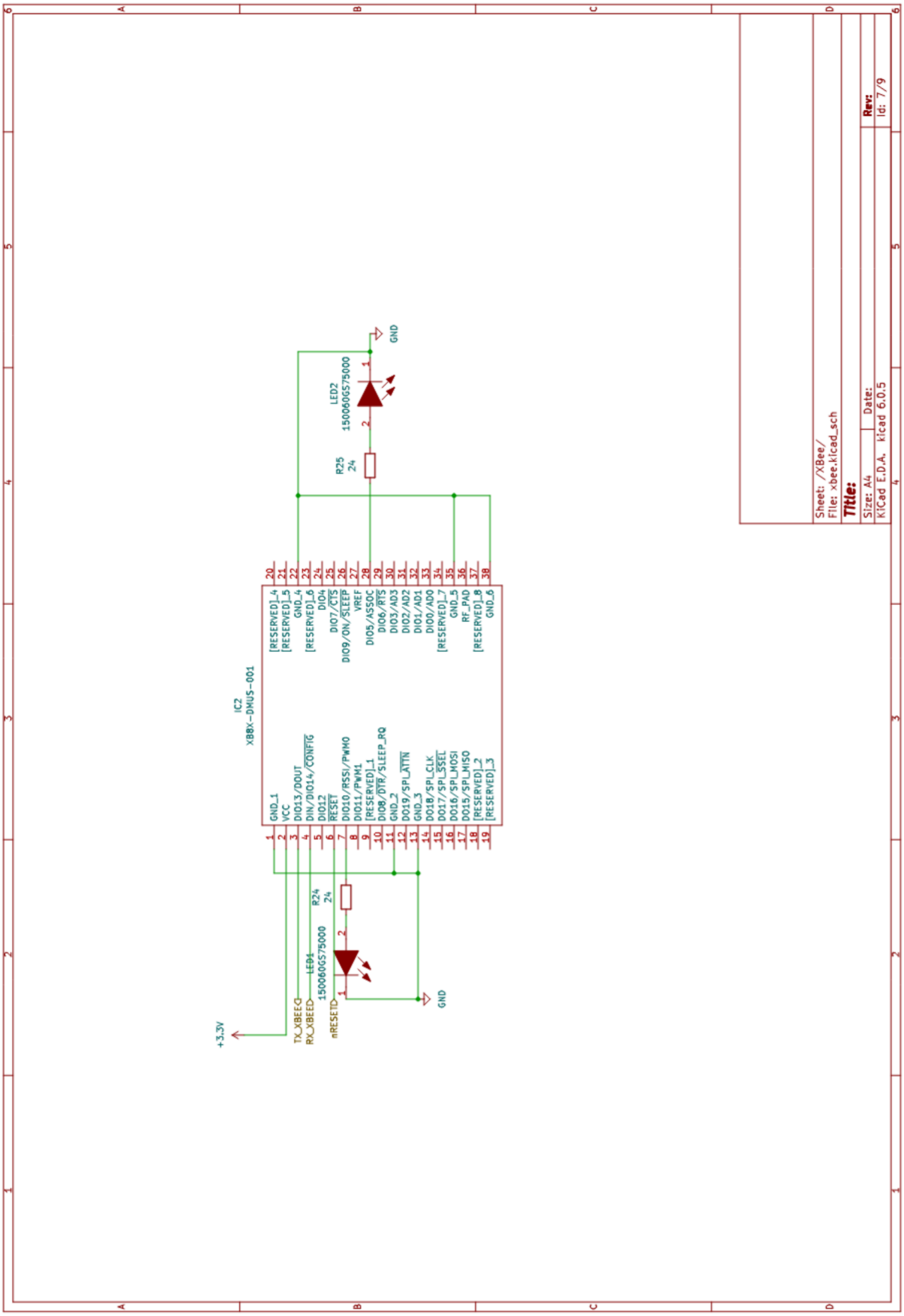
Figure A.5: Wildhorn sensor PCB BOM

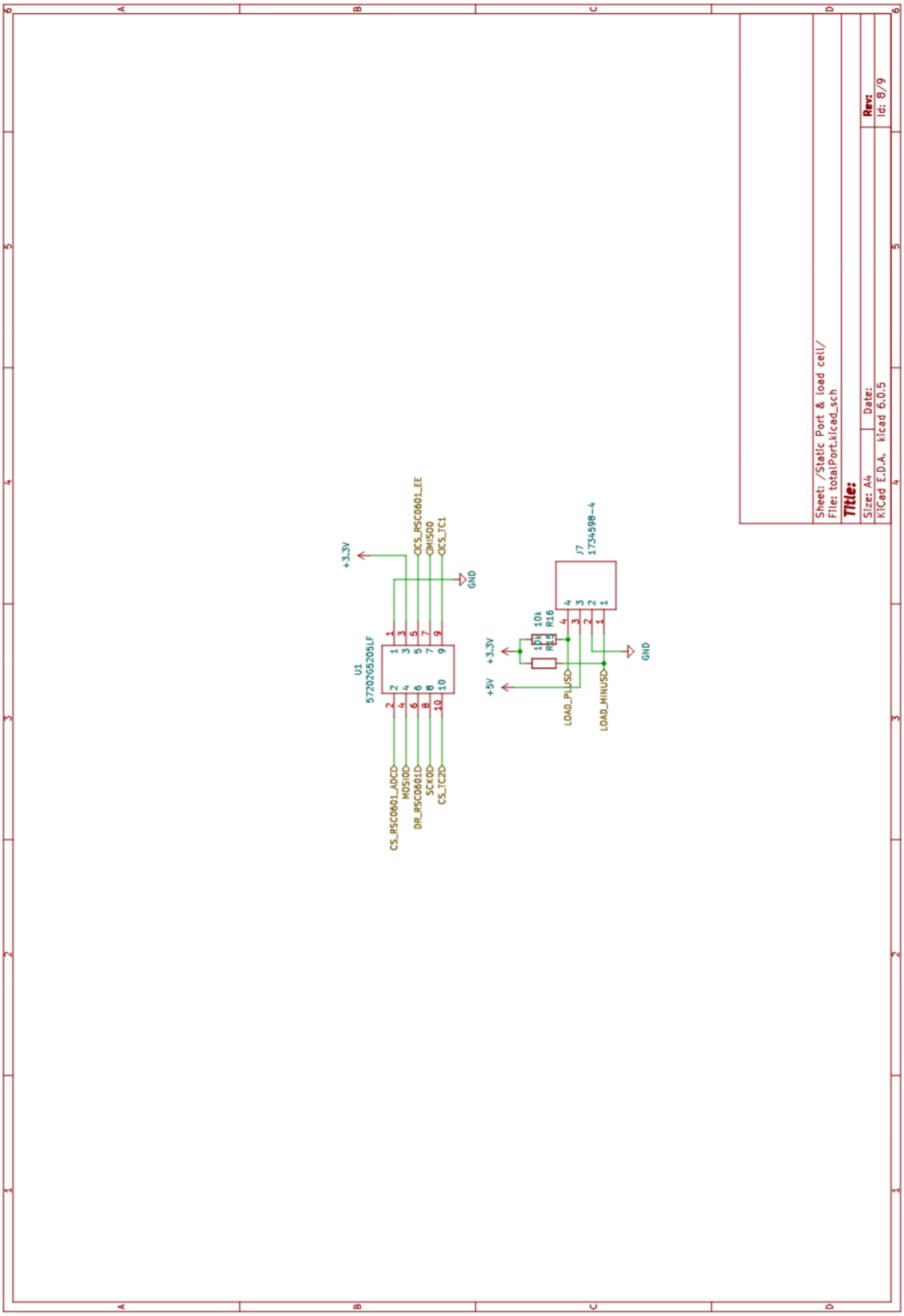

Teensy
U4
Teensy3.5
CustomLib:Teensy35_36_exterior_pins_only
RTC_&_reset
J2
0878980304
U5
3030
XBee
Fichier: xbee.kicad_sch
Static Port & load cell
Fichier: totalPort.kicad_sch
Total Port
Fichier: staticPort.kicad_sch
Gps
Fichier: gps.kicad_sch
Power
Fichier: power.kicad_sch
Wildhorn
Fichier: wildhorn.kicad_sch
I/O
Fichier: IO.kicad_sch
INS
Fichier: INS.kicad_sch
Sheet: /
File: PCB_v1.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 1/9

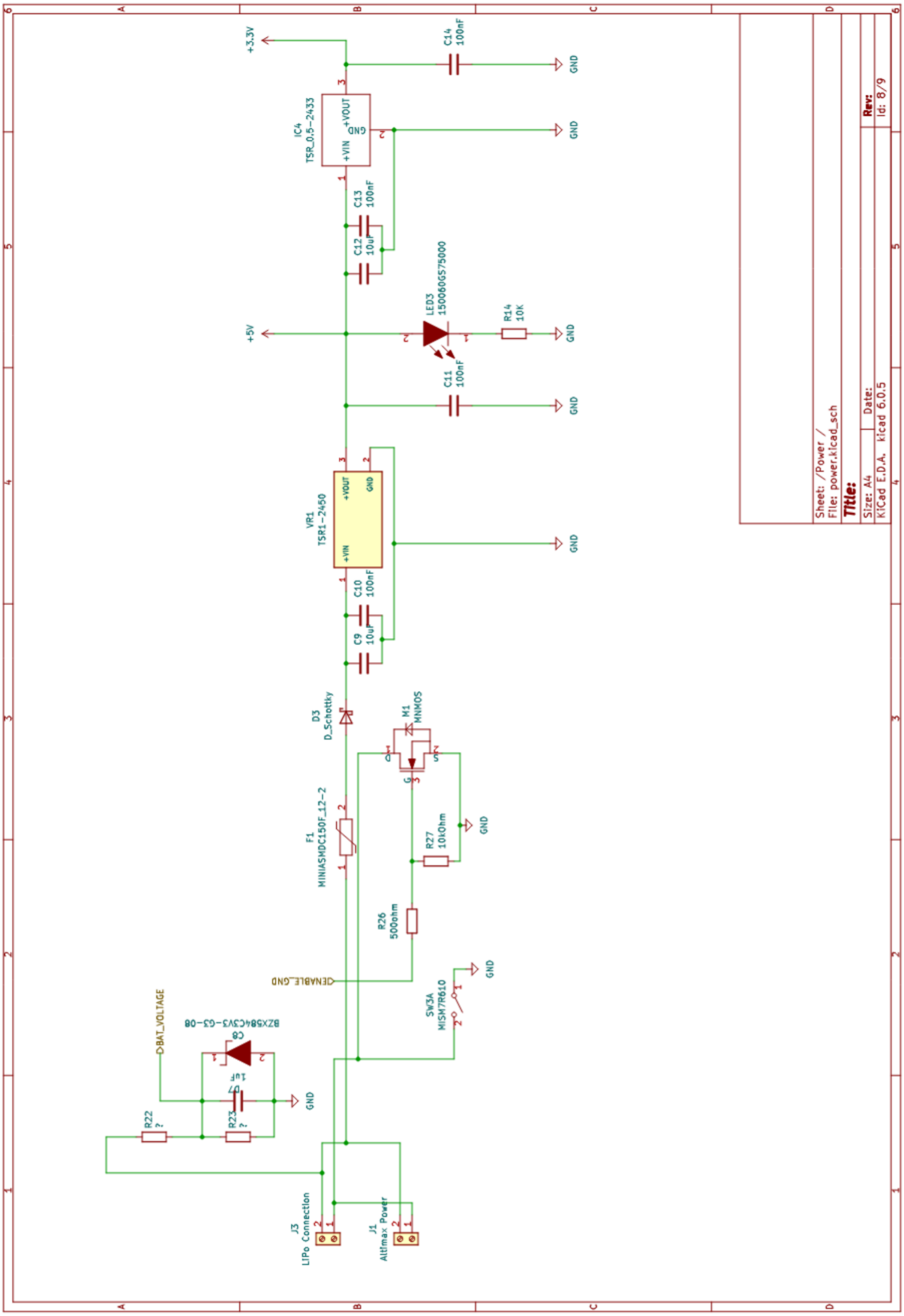

GND
L3
SMI-1027-T-5V-R
BUZZER_INPUT
+3.3V
GND
U6
switch
U7
switch
C2
100nF
+3V3
C1
100nF
R1
10kΩ
GND
Button1
nReset
D1
White LED
D2
Red LED
R3
100Ω
R5
24Ω
R6
10kΩ
R4
10kΩ
GND
LED1
LED2
AUX2
AUX1
R10
GND
J5
57202G5203LF
R2
10k
GND
+3.3V
AUX3
Sheet: /I/O/
File: IO.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 4/9

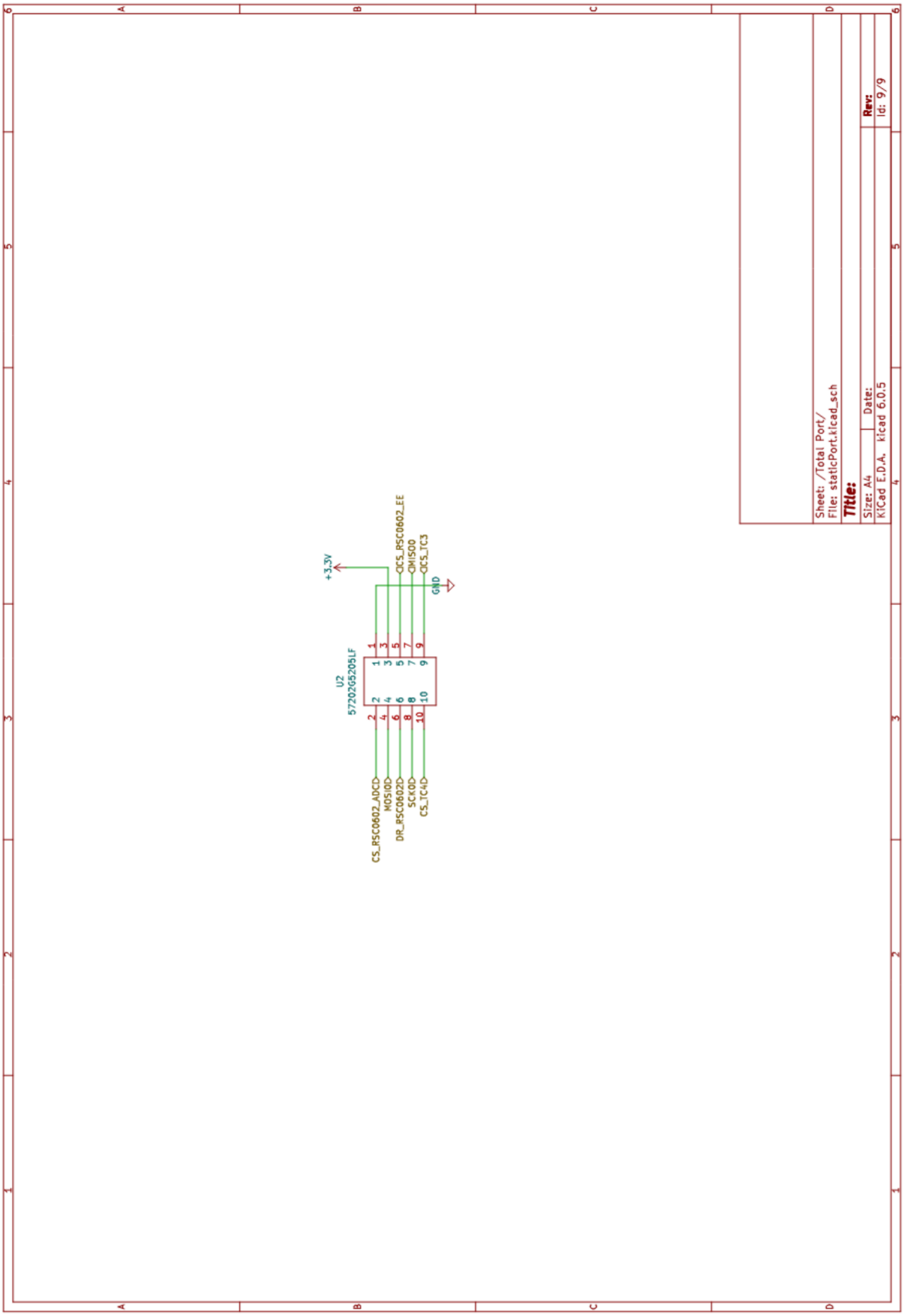

U10
AIS1120SXTR
SCL
SDI
SDO
CS
MP
VDD
VREG
GND
C7
100nF +/- 10% 10V
C6
1uF +/- 10% 10V
C5
100nF
R13
10kΩ
R12
10kΩ
R11
10kΩ
R20
10kΩ
+3.3V
GND
SCKO
MOSIO
MISOO
ADIS16470
U9
DNC
SYNC
VDD
GND
DIN
DOUT
SCLK
DR
CS
RST
MOSIO
MISOO
SCKO
CS_AIS1120
CS_ADIS16470
DR_ADIS16470
SYNC_ADIS16470
RST_ADIS16470
Sheet: /INS/
File: INS.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 5/9

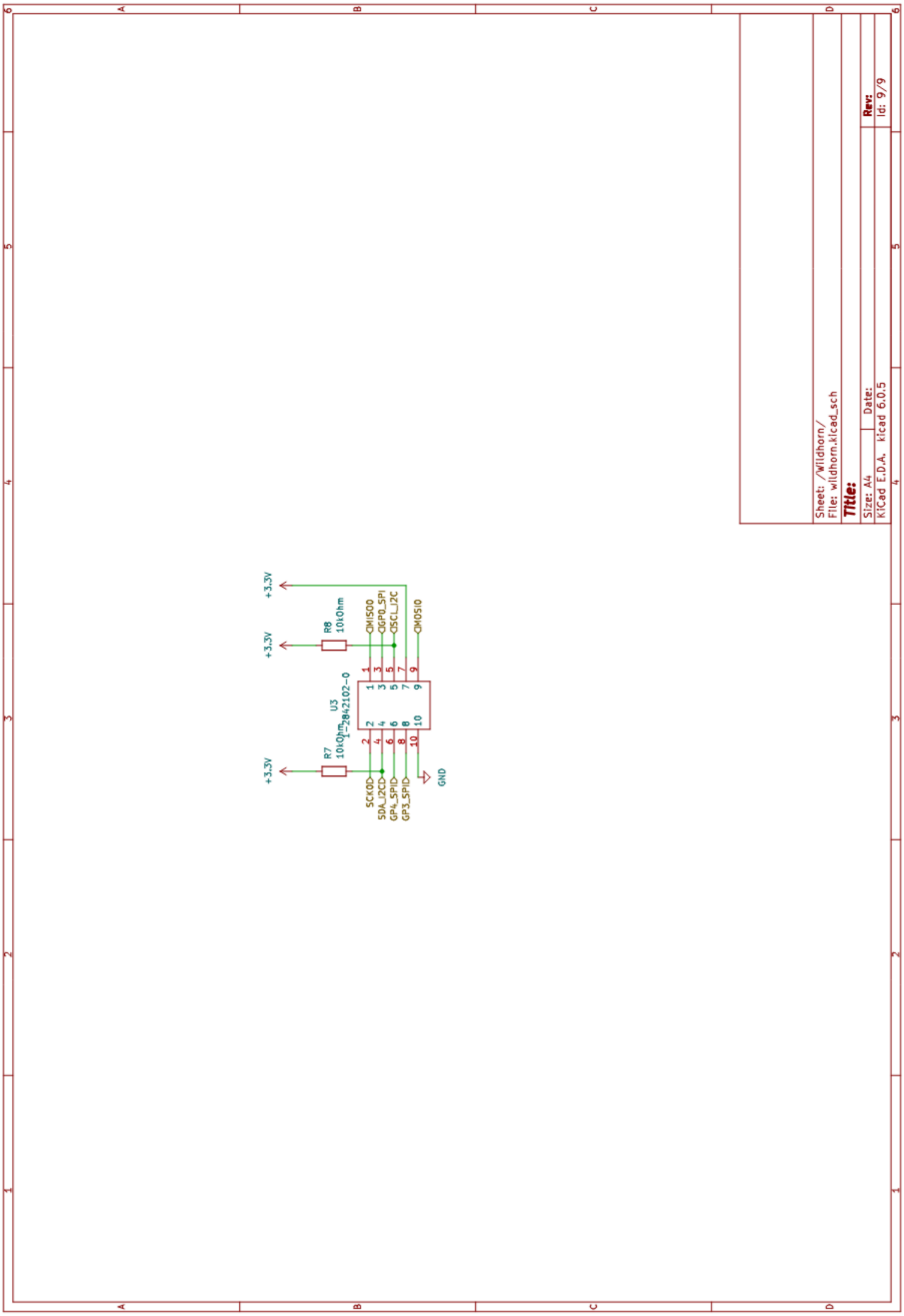

nRESET
RTC_SUPPLY
TIMEPULSE_MAX7C
RX_MAX7C
TX_MAX7C
+3.3V
GND
C4
10uF
C3
100nF
IC1
MAX-7C-0
RESET_N
VCC
VCC_IO
V_BCKP
EXTINT
TIMEPULSE
RXD
TXD
GND_1
GND_2
RF_IN
GND_3
ANT_ON
VCC_RF
RESERVED_1
SDA
SCL
RESERVED_2
J4
CONUFL001-SMD-T
SIG
GND_1
GND_2
Sheet: /Gps/
File: gps.kicad_sch
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 6.0.5
Id: 6/9

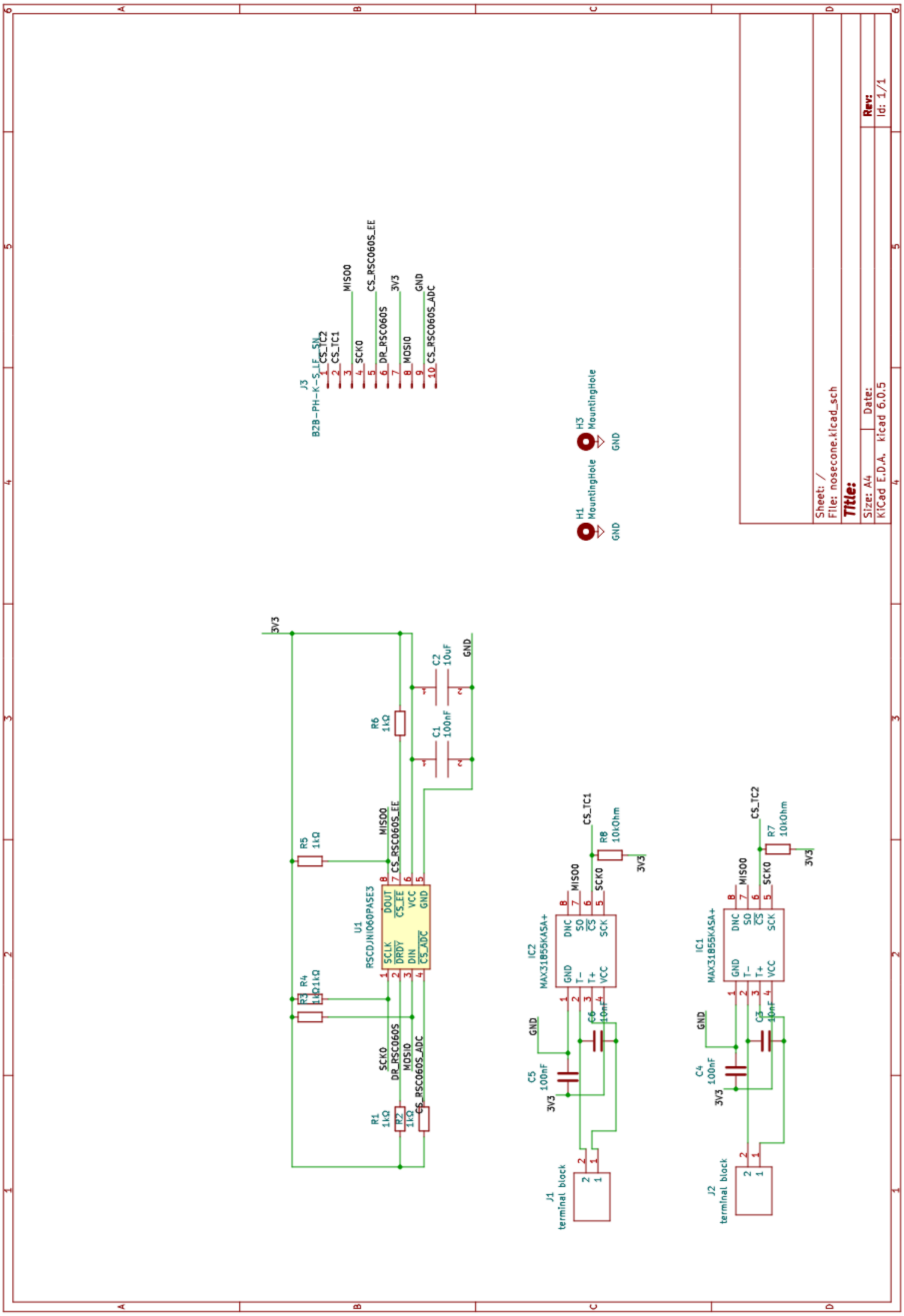

GND
LED2
150060GS75000
R25
24
IC2
XBBX-DMUS-001
GND_1
VCC
DIO13/DOUT
DIN/DIO14/CONFIG
DIO12
RESET
DIO10/RSSI/PWM0
DIO11/PWM1
[RESERVED]_1
DIO8/DTR/SLEEP_RQ
GND_2
DIO19/SPI_ATTN
GND_3
DIO18/SPI_CLK
DIO17/SPI_SSEL
DIO16/SPI_MOSI
DIO15/SPI_MISO
[RESERVED]_2
[RESERVED]_3
[RESERVED]_4
[RESERVED]_5
GND_4
[RESERVED]_6
DIO4
DIO7/CTS
DIO9/ON/SLEEP
VREF
DIO5/ASSOC
DIO6/RTS
DIO3/AD3
DIO2/AD2
DIO1/AD1
DIO0/AD0
[RESERVED]_7
GND_5
RF_PAD
[RESERVED]_8
GND_6
R24
24
LED1
150060GS75000
GND
+3.3V
TX_XBEE
RX_XBEE
nRESET
Sheet: /XBee/
File: xbee.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 7/9

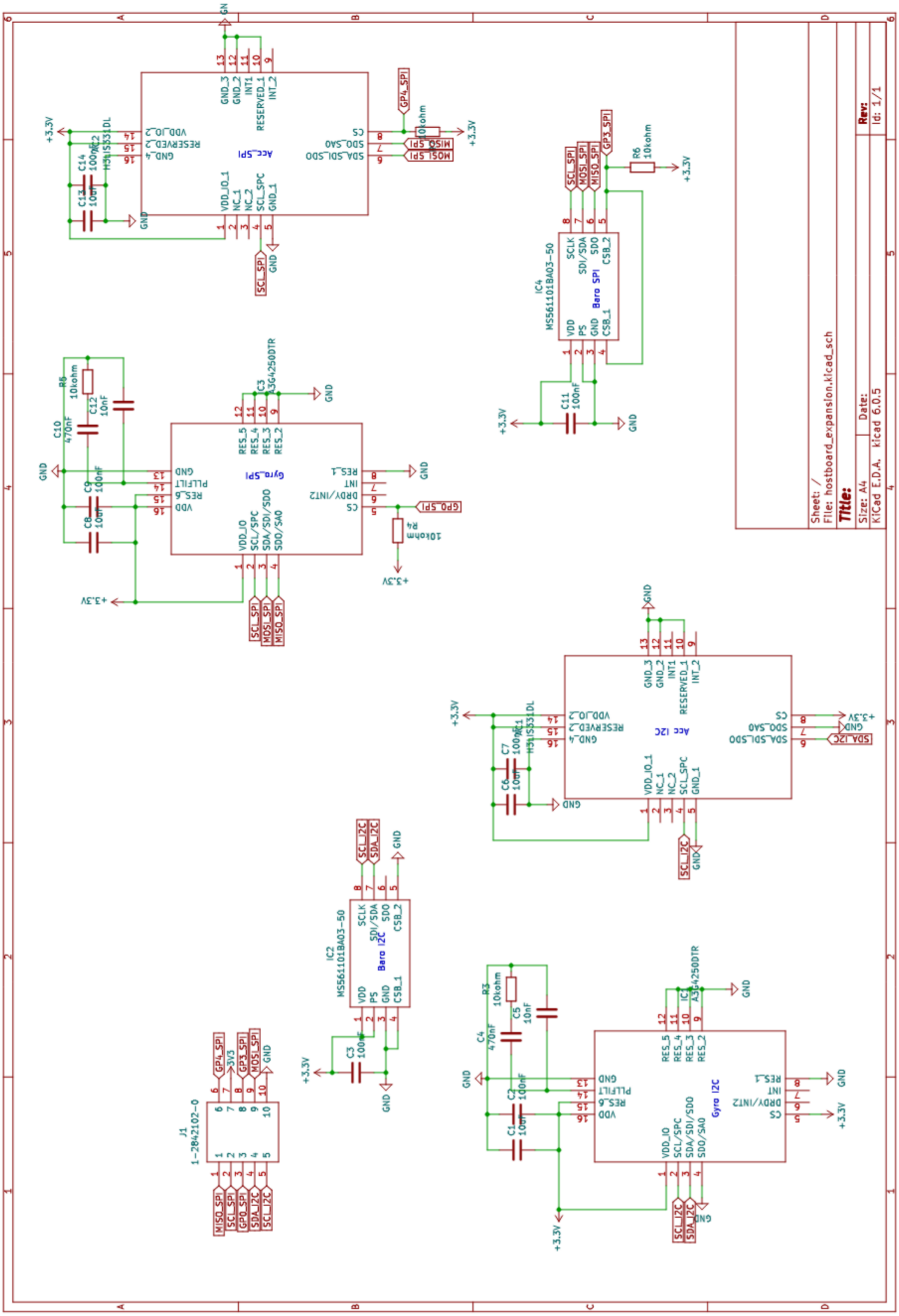

Sheet: /Static Port & load cell/
File: totalPort.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 8/9
U1
57202G52S05LF
J7
1734598-4
R15
R16
10k
+3.3V
+5V
GND
CS_RSC0601_EE
MISO0
CS_TC1
CS_RSC0601_ADC
MOSI0
DR_RSC0601
SCK0
CS_TC2
LOAD_PLUS
LOAD_MINUS

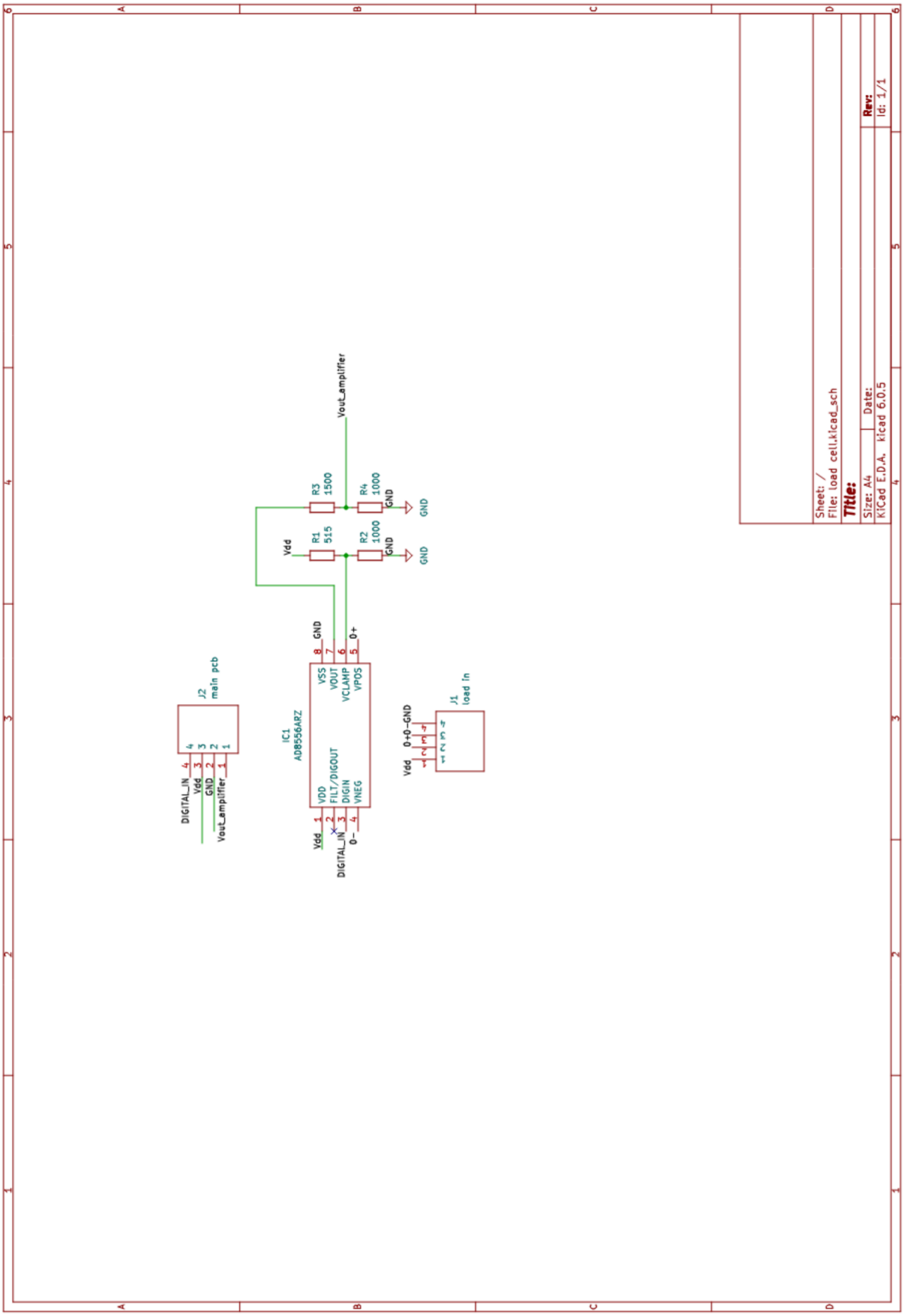

+3.3V
C14
100nF
GND
IC4
TSR_0.5-2433
+VIN
+VOUT
GND
C13
100nF
C12
10uF
LED3
150060GS75000
R14
10K
+5V
C11
100nF
VR1
TSR1-2450
C10
100nF
C9
10uF
D3
D_Schottky
M1
MNMOS
F1
MINIASMDC150F_12-2
R27
10kOhm
R26
500ohm
ENABLE_GND
SW3A
MISM7R610
BAT_VOLTAGE
BZX584C3V3-G3-08
C8
C7
1uF
R22
?
R23
?
J3
LiPo Connection
J1
Altimax Power
Sheet: /Power /
File: power.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 8/9

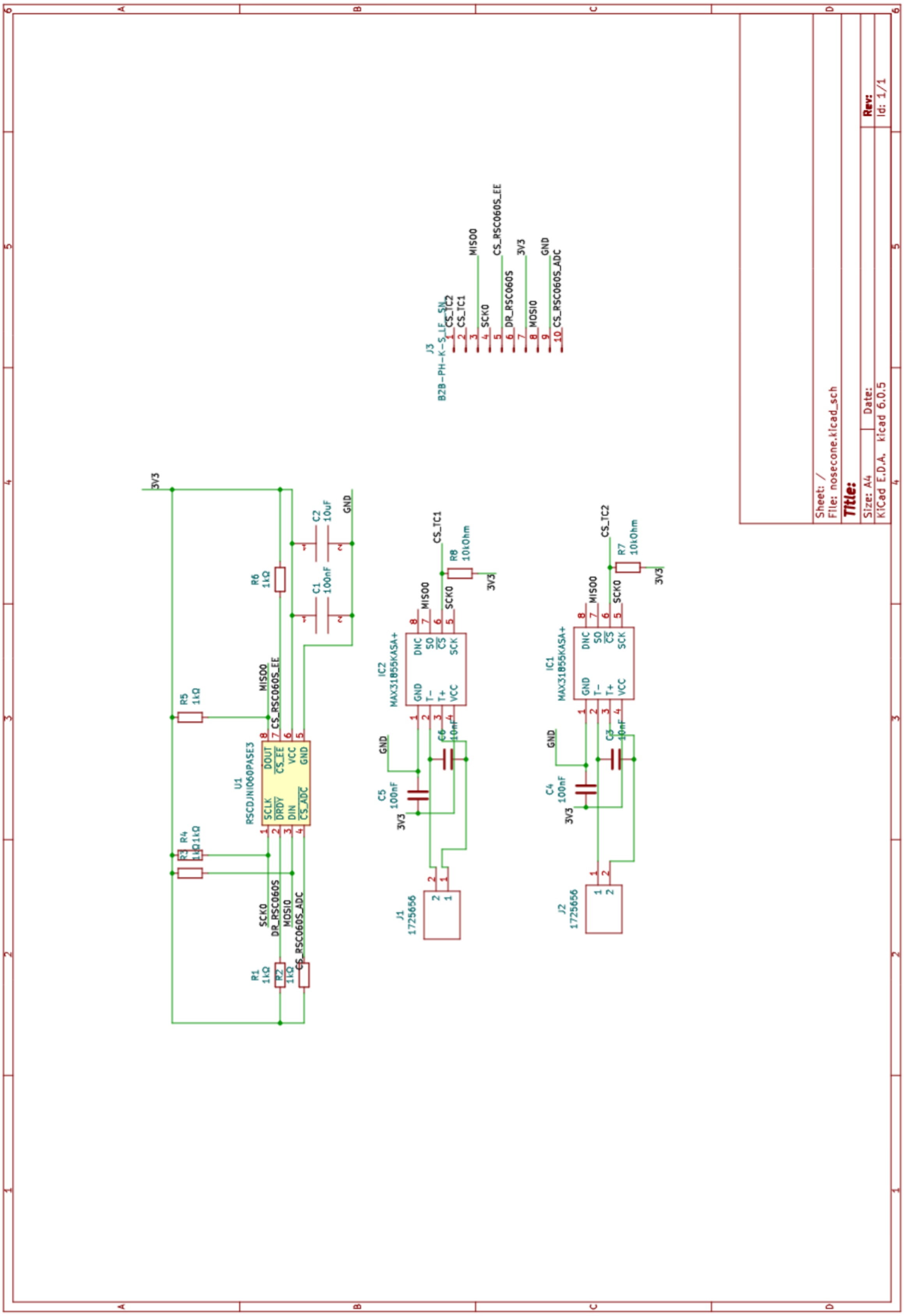

Sheet: /Total Port/
File: staticPort.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 9/9
U2
57202G5205LF
+3.3V
GND
CS_RSC0602_EE
MISO
CS_TC3
CS_RSC0602_ADC
MOSI
DR_RSC0602
SCK
CS_TC4

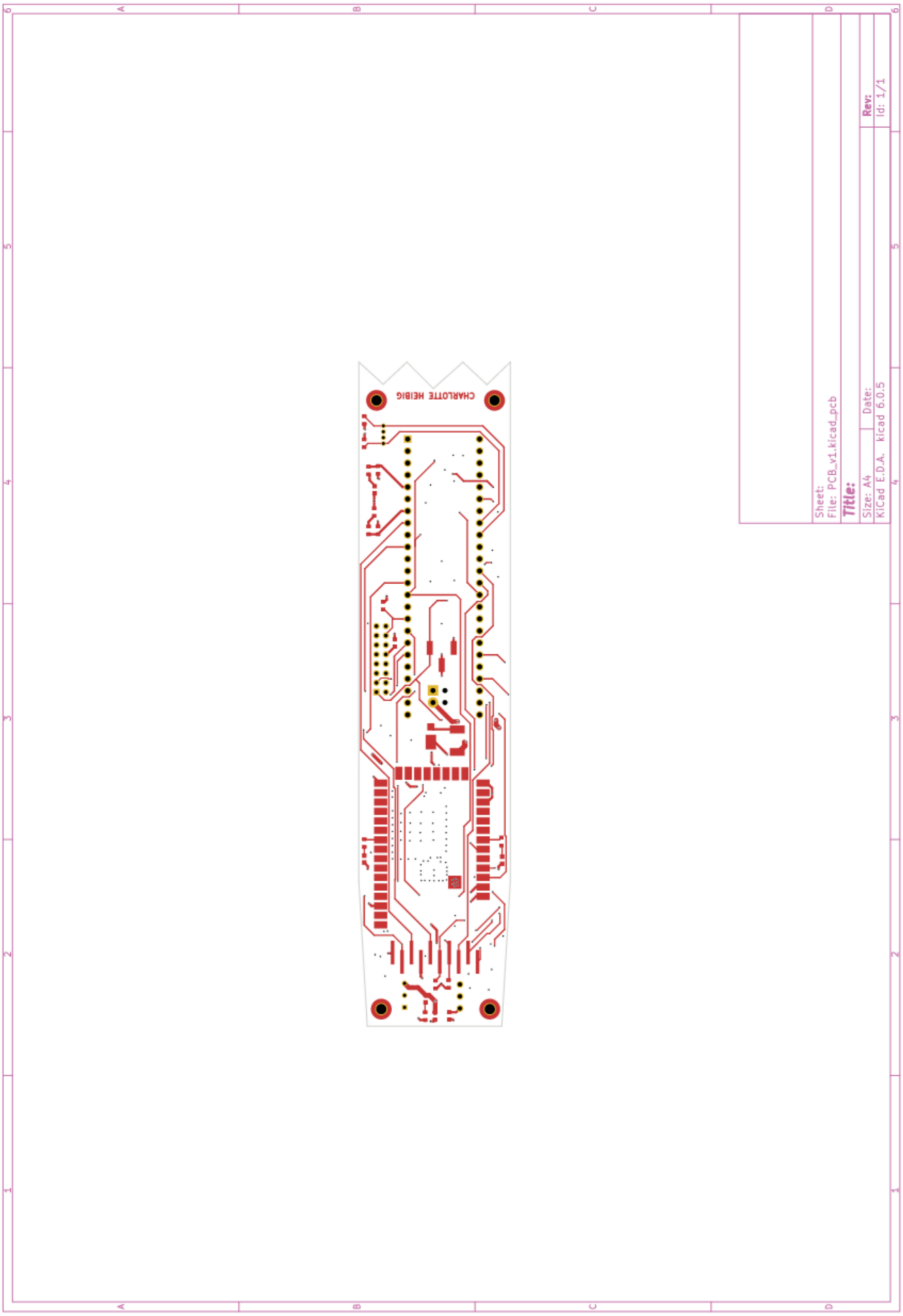

+3.3V
R8
10kOhm
MISO0
GP0_SPI
SCL_I2C
MOSI0
U3
1-2842102-0
R7
10kOhm
SCK0
SDA_I2C
GP4_SPI
GP3_SPI
GND
Sheet: /Wildhorn/
File: wildhorn.kicad_sch
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 6.0.5
Id: 9/9

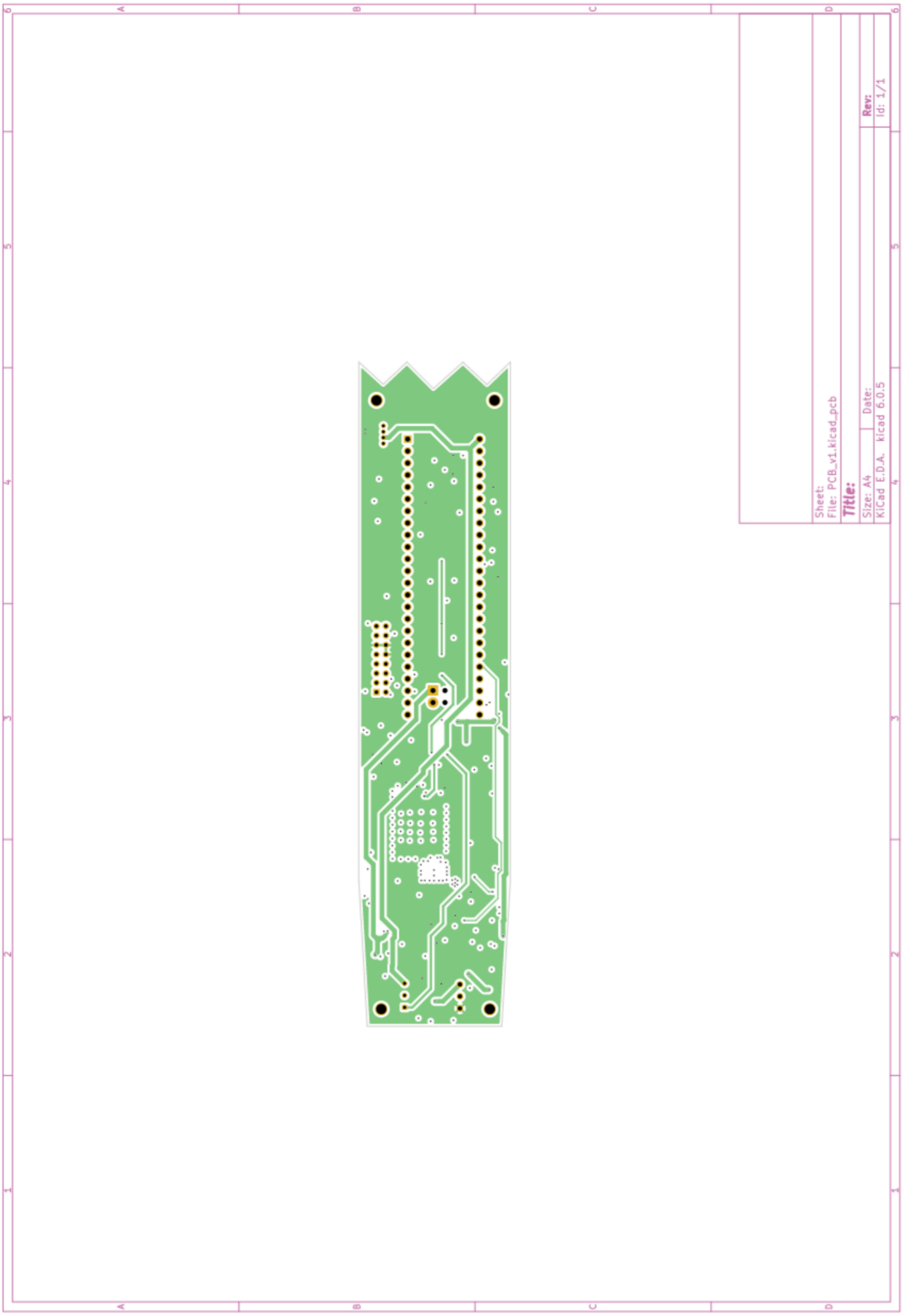

Sheet: /
File: nosecone.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 1/1
J3
B2B-PH-K-S_LF_SN
CS_TC2
CS_TC1
SCK0
DR_RSC060S
MOSI0
CS_RSC060S_ADC
MISO0
CS_RSC060S_EE
3V3
GND
H1
MountingHole
H3
MountingHole
U1
RSCDJNI060PASE3
SCLK
DRDY
DIN
CS_ADC
DOUT
CS_EE
VCC
R1
1kΩ
R2
1kΩ
R4
1kΩ
R5
1kΩ
R6
1kΩ
C1
100nF
C2
10uF
IC2
MAX31855KASA+
IC1
MAX31855KASA+
DNC
SO
CS
SCK
T-
T+
R8
10kOhm
R7
10kOhm
C5
100nF
C6
10nF
C4
100nF
C3
10nF
J1
terminal block
J2
terminal block

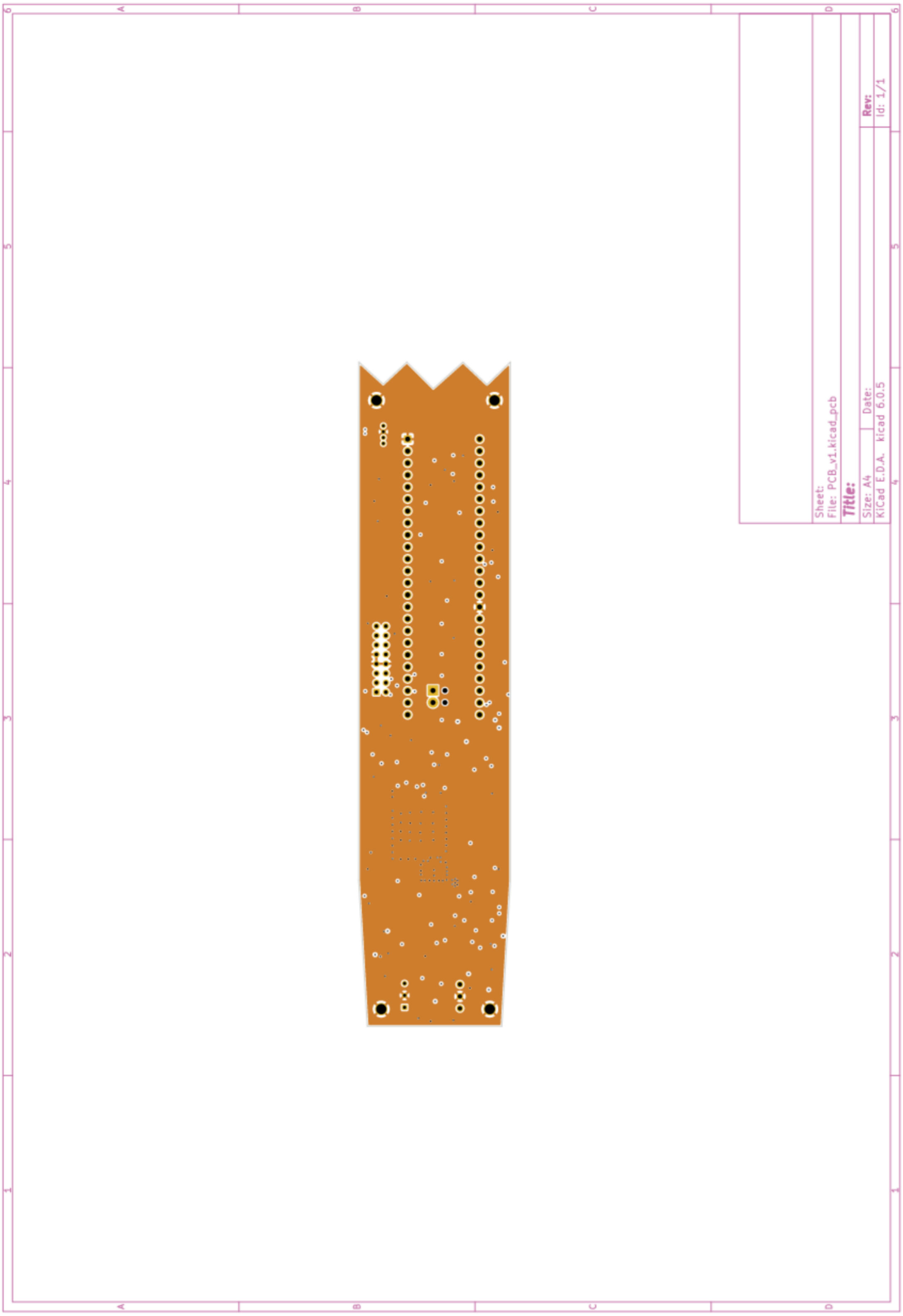

Sheet: /
File: hostboard_expansion.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 1/1

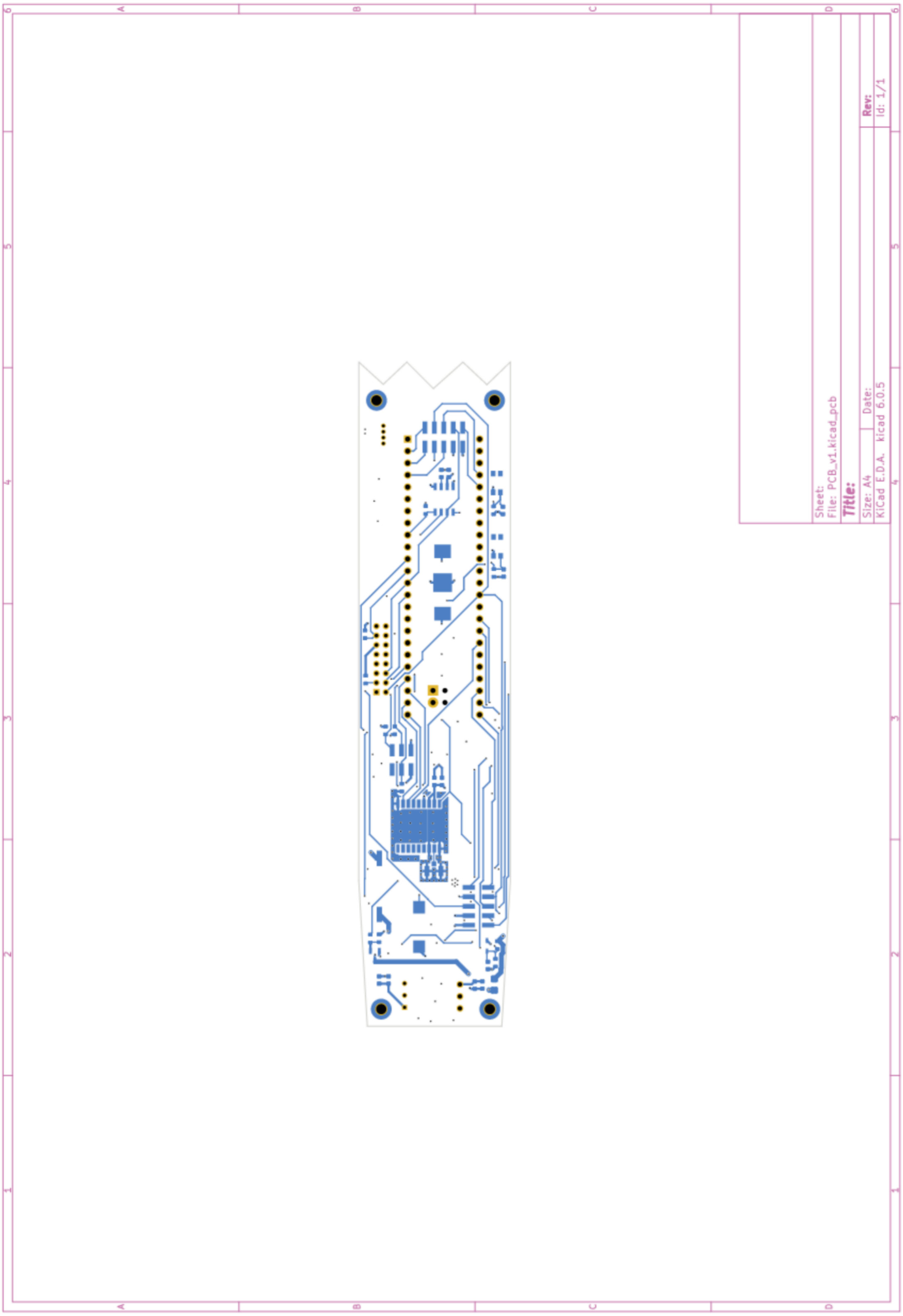

Vout_amplifier
R3
1500
R4
1000
GND
R1
515
R2
1000
Vdd
GND
IC1
AD8556ARZ
VSS
VOUT
VCLAMP
VPOS
VDD
FILT/DIGOUT
DIGIN
VNEG
DIGITAL_IN
0+
0-
J2
main pcb
Vout_amplifier
J1
load in
Sheet: /
File: load cell.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 1/1


U1
RSCDJNI060PASE3
IC2
MAX31855KASA+
IC1
MAX31855KASA+
J1
1725656
J2
1725656
J3
B2B-PH-K-S_LF_SN
Sheet: /
File: nosecone.kicad_sch
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 1/1

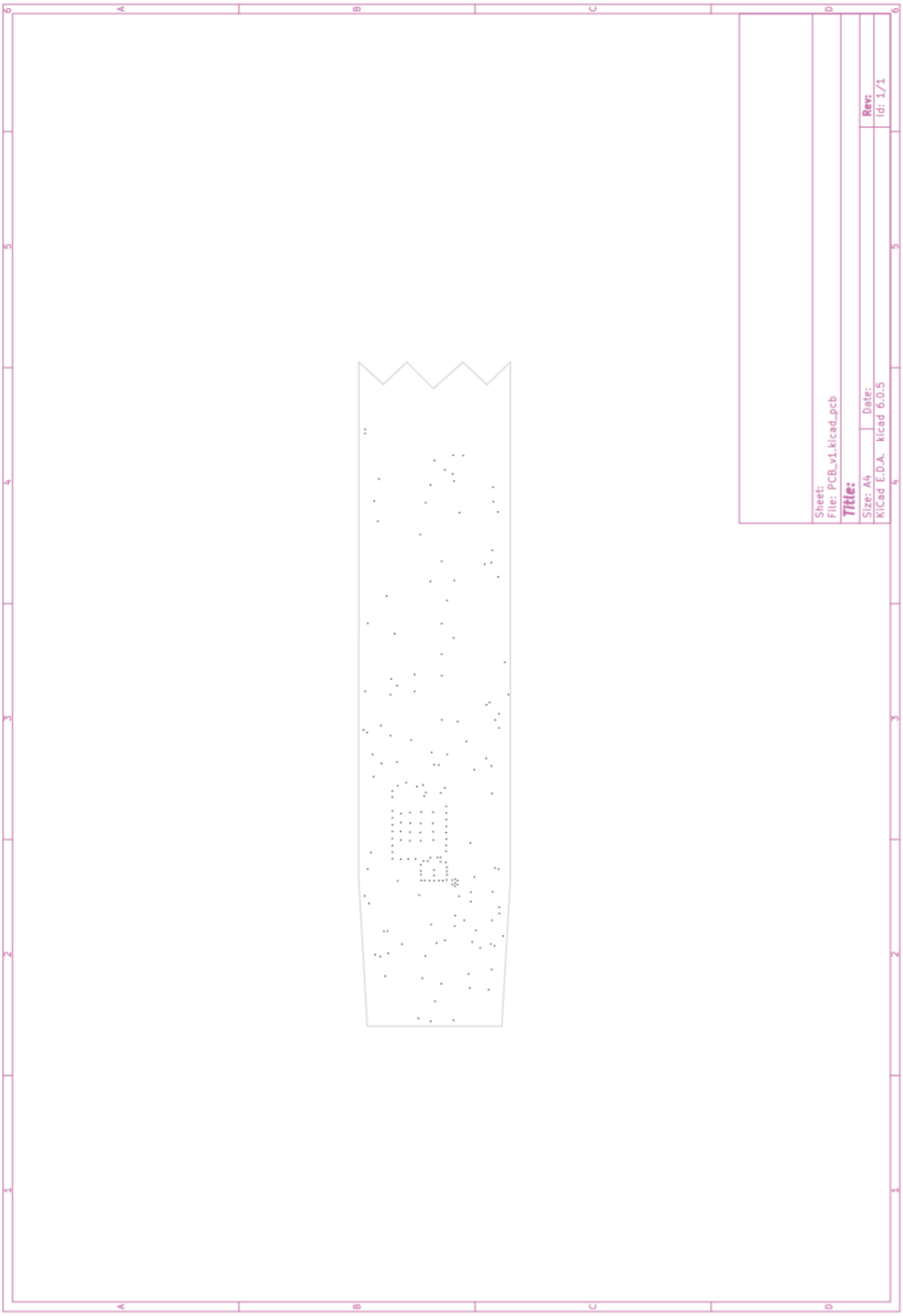

CHARLOTTE HEIBIG
Sheet:
File: PCB_v1.kicad_pcb
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 6.0.5
Id: 1/1


Sheet:
File: PCB_v1.kicad_pcb
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 6.0.5
Id: 1/1


Sheet:
File: PCB_v1.kicad_pcb
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 1/1



Sheet:
File: PCB_v1.kicad_pcb
Title:
Size: A4
Date:
Rev:
KiCad E.D.A. kicad 6.0.5
Id: 1/1


Sheet:
File: PCB_v1.kicad_pcb
Title:
Size: A4
Date:
KiCad E.D.A. kicad 6.0.5
Rev:
Id: 1/1