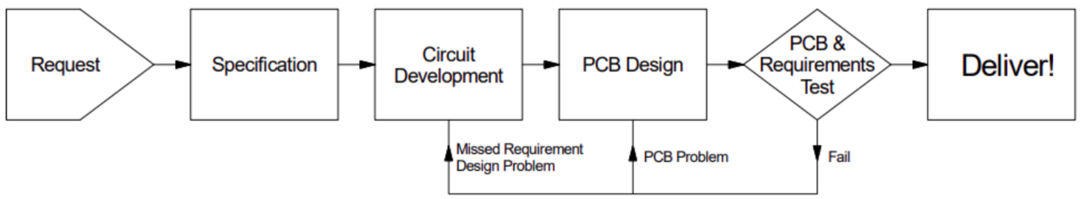
Constructor University Bremen

PCB Lab

Author:Prince Lee Muhera

- **Introduction**

  This internship aims to provide insight into the design flow of electrical devices. The following describes various steps and some aspects to consider when turning a circuit idea into a real device. The entire process is illustrated by an example below:



1. The design starts with a request.

2. A product specification has to be developed. This is the base for the circuit design and has to be as detailed and accurate as possible!
3. Circuit development is based on the product specification.

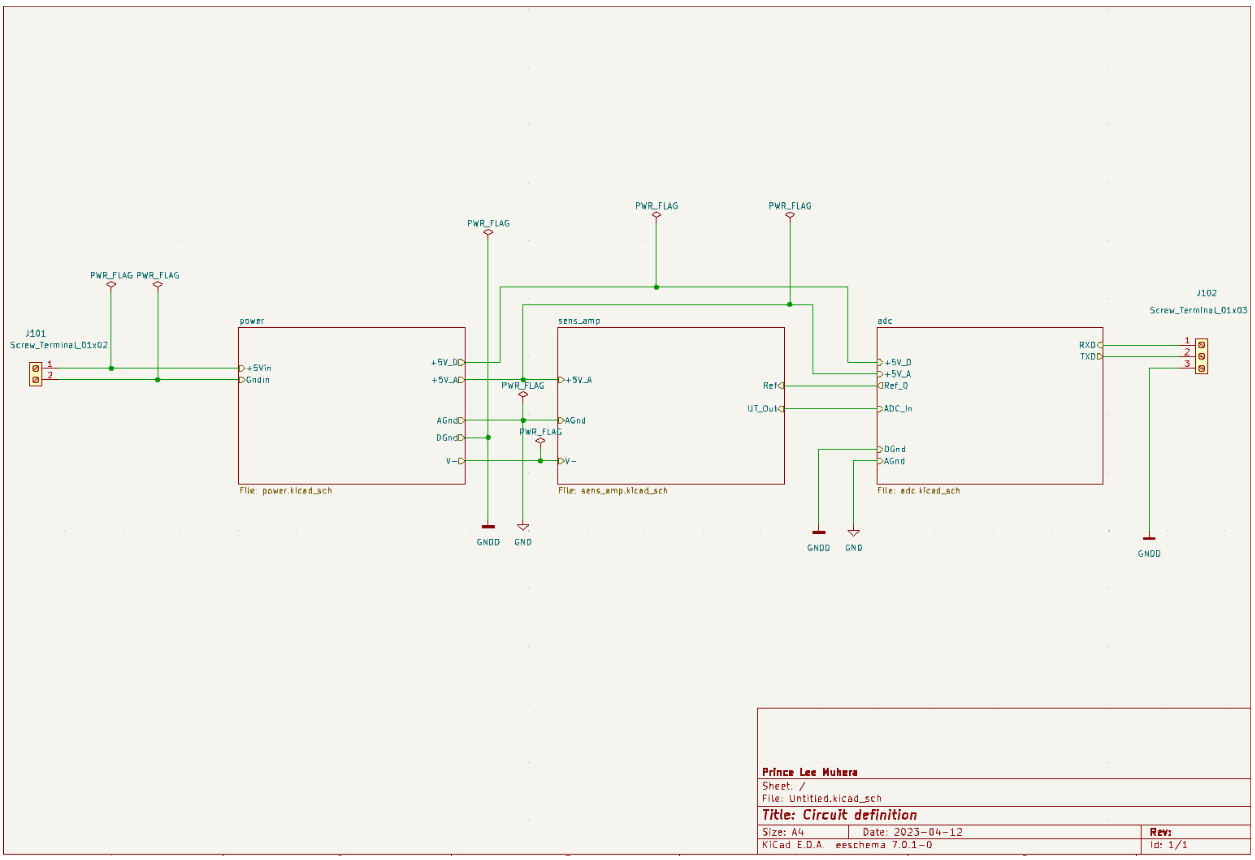
   - Define, develop, and simulate the circuit(s).
   - Specify the needed components.

4. Circuit design.
   - Finalize schematics, fix mechanical setup, and PCB layout
5. Final test of the device. Changes or redesign if required.

- **Specification Sheet**
  - Function:
    Develop a circuit which measures temperature and transmits the result via RS232. Temperature range should be from -10° to $+50^{\circ}$C.
  - Accuracy: ±(5%rdg + 4dgt)
  - Sample rate, stored history: 2Val/min, Values of last two hours
  - Use a 5V ± 10% supply. Maximum power consumption 200mW.

- ## Circuit definition

  It is a good practice after the first idea to split the circuit into functional blocks. In our design we need a power supply, some analog circuitry to detect temperature, a digital part with analog to digital conversion, and data processing and transmission.



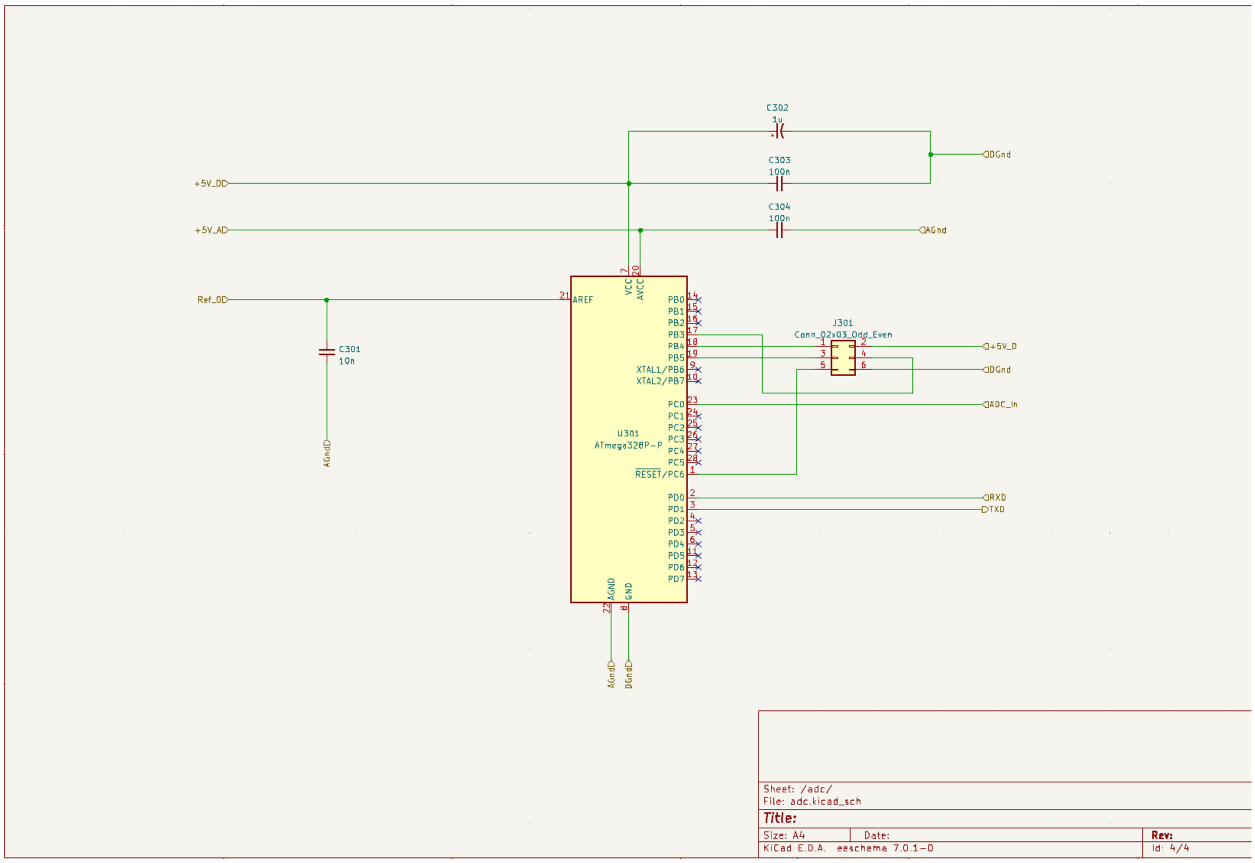
  Power - Power supply +5V for analog and digital part.

  Analog - Sens Amp (Sensor & Amplifier)

  Digital - Combined ADC and I2C

- ## Specification and Realization of the Digital Part

The digital part requires an ADC , timer, serial port, and memory for data. A single microcontroller is used that contains all the required components. Example: Atmega168P (or Atmega88P) processor used on Arduino boards. The digital part is a very economical design. All you need is a microcontroller, a programming head, and a blocking / filtering capacitor for the reference and digital supply voltages.



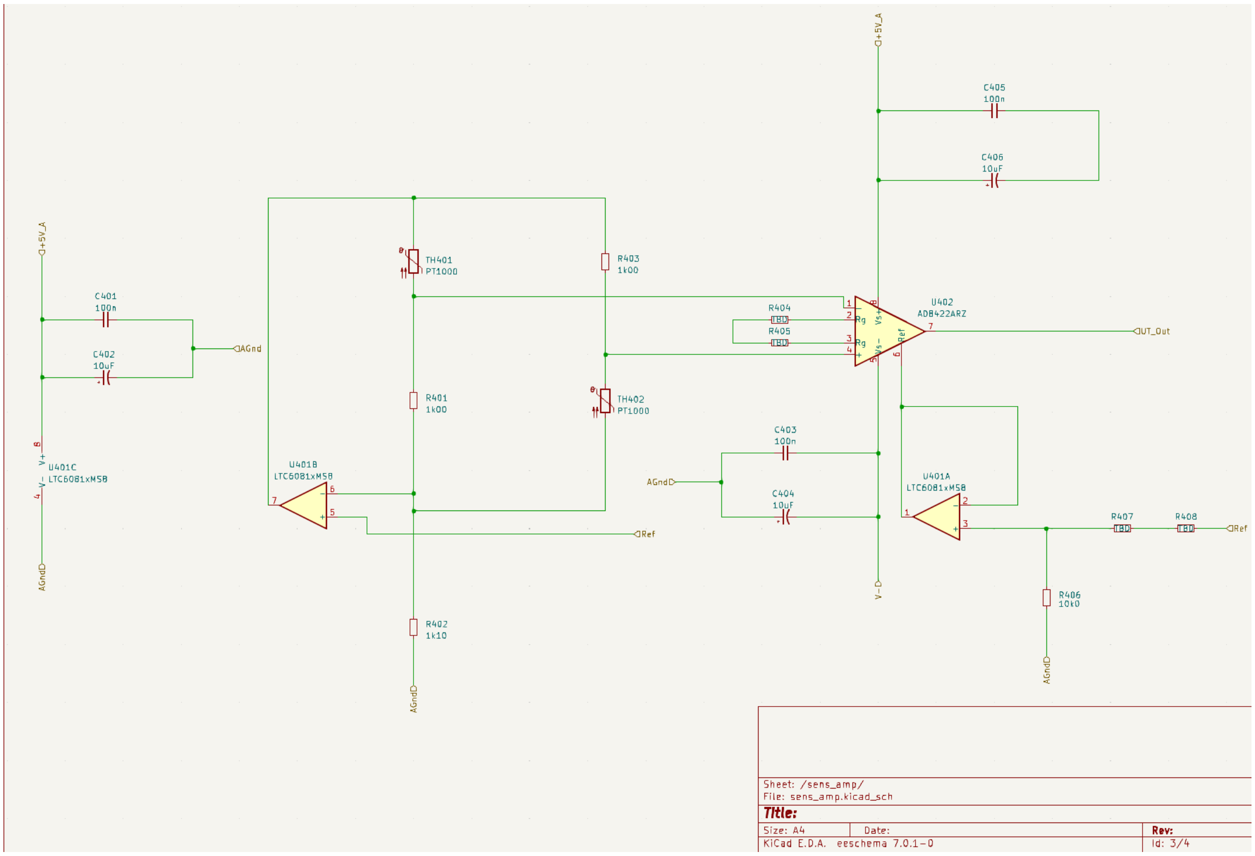
<u>Requirements</u>

- Timer - available, nearly any timing possible
- Analog digital convertor
- It is including an easy controllable 10 bit ADC. One sample every 30s is no problem.
  At one degree it increases the error to ≈ 10%. Acceptable, keeps the system cheap!!
- Memory for data
- 1024 Byte RAM is available, needed 2V al/min ∗ 2 ∗ 60min = 240V also

- Since a value has 2 Byte 480Bytes are needed. That leaves enough RAM for program
use!
- Serial Interface - RS232 interface is available.
- The MC includes a 1.1V reference voltage source we can use for the analog part (1.074mV/bit).

- **Specification and Realization of the Analog Part**

  Digital components require a reference voltage, sensors, and amplifiers. Other components such as the following can also be used in the circuit: The output of the controller is too weak and the reference voltage needs to be boosted. A simple solution is to use a precision op amp with a non-inverting configuration as a buffer. The next device is the PT1000 or 2 as a Wheatstone bridge temperature sensor for detecting changes in resistance. An instrumentation amplifier used as a differential amplifier to boost the output of a bridge circuit.

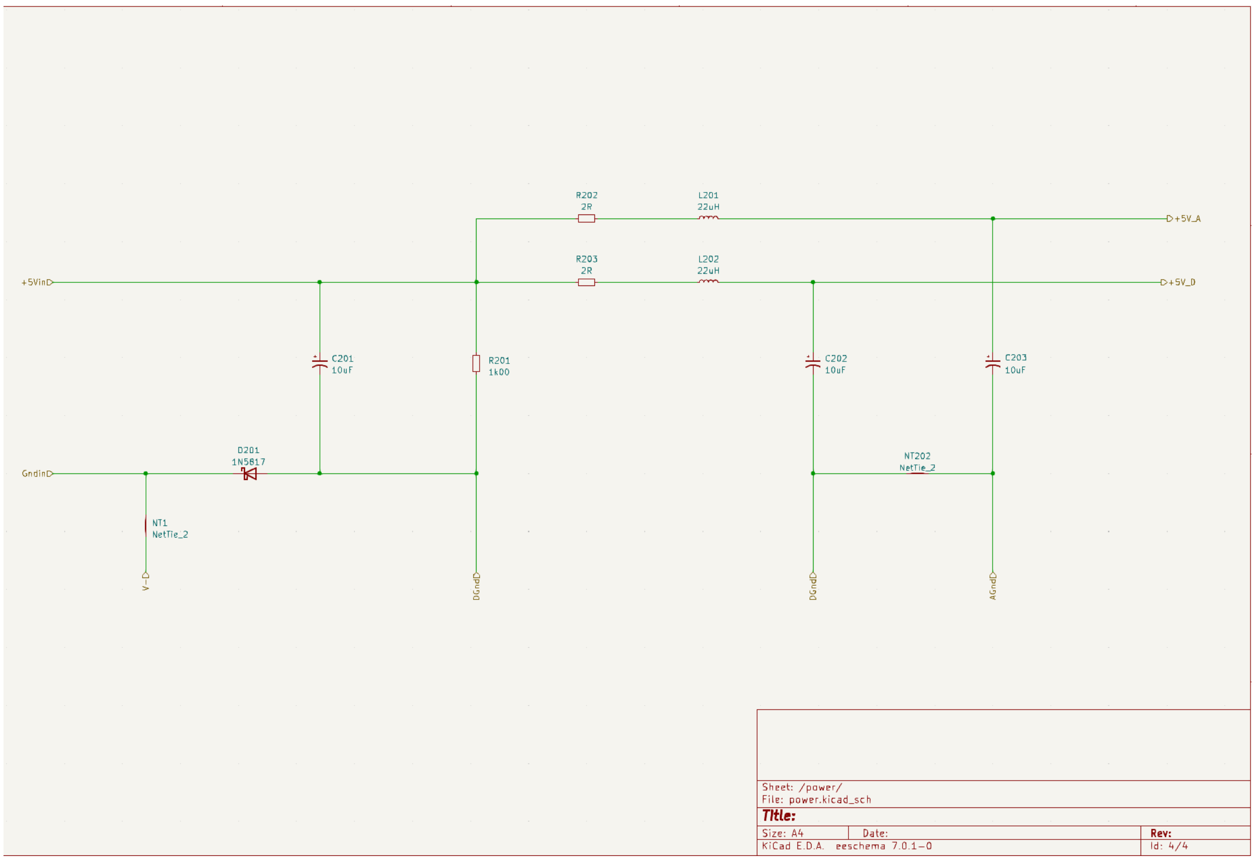


The reference voltage needs amplification.The reference voltage from the controller is too weak. Also the reference pin of the instrumentation amplifier needs a low impedance source. Easy solution is to use a precision OP-Amp in non-inverting configuration as buffer. One or two PT1000 as temperature sensors .. inside a Wheatstone Bridge to detect the change in resistance.

An instrumentation amplifier used as differential amplifier to amplify the output of the bridge and a circuit.

- ## Specification and Realization of the Power Part

For the Power part, 5V is the requested supply voltage of the circuit. Build up a circuit all the power requirement of all the active components needs to be determined and hence the overall power consumption needs to be taken care as well. All the elements in the power block function properly for voltage at least up to 5V. The controller needs up to 10 mA in worst case. While, the current for the Wheatstone reference is about 1mA. Instrumentation amplifiers require a small negative supply voltage to reach 0V at the output. The power supply must be 5V and all components can use 5V, so no additional regulators or converters are needed. The main part is just an LC filter to reduce ripple / noise from the power supply. This is because instrumentation amplifiers require a small negative supply voltage to operate up to 0V at the output. Therefore, GND is shifted by about 0.3V with a Schottky diode.



- To find a circuit we first have to determine the power requirements of all active components and the overall power consumption!
- All elements can work with at least up to 5V . Now for every functional block we have to estimate the current then we add up.
- The controller needs up to 10mA in worst case.
- The two amplifiers up to 1mA supply current.
- The current for the Wheatstone reference is about 1mA.
- To be save the external supply should deliver 5 V ± 10% and ≈ 20 mA.
- Since the supply should have 5V and all components can use 5V we do not need an extra regulator or converter. The main part is only a LC filter to reduce ripple/noise from the power source!! Since the instrumentation amplifier needs a

small negative supply voltage to operate down to 0V at the output the GND is shifted by ≈ 0.3V using a Schottky diode.

- ## Engineering Part

The next step is to check the structure and fill in the dimensions. You need to achieve the desired characteristics. You need to determine the application limits of the component and the required margins of error. Depending on the problem, searching for the component may take some time. You need to read the datasheet to find the component. The circuit should be simulated or built on a breadboard. You need to estimate the error over the entire operating range. The final step is to define the housing of the various components to prepare for the PCB design. After simulation and testing, you can use the following components:

1. Define - Digital Part

- Microcontroller Atmega168P, Package DIP-28 Pin
- polarized C - AVX TAJA105M020R
  Chip capacitor, Tantalum 20 V, Size A (EIA Code 3216)
  not polarized C - AVX 08055C104J4T2A
  Chip capacitor, Ceramic 50V , size 0805
- Connector J301 - 2 Rows 6 Pins, Grid 2.54, through hole
  Standard connector for Atmega programming adaptors
  Connector J101
  J102 - Screw terminal 2 Pin, Grid 2.54

2. Define - Power

- Resistor - Vishhay CRCW08051K00FKEA, Standard Thick Film Chip Resistor, Size 0805
- polarized C - KEMET A700V106M016ATE045
  Chip capacitor, Size 7343, Aluminium 16V
  not polarized C - AVX 08055C104J4T2A
  Chip capacitor, size 0805, Ceramic 50V
- Inductors - Murata LQH32MN220J23L or similar, low resistive, high Q Chip inductor, size 1210
- Diode - Shottky 1N5817, Package DO-41

3. Define - Sens Amp

- Bridge R - Metal Film Vishhay Y16241K00000T9R
  Ultra High Precision Foil Wraparound Chip Resistor, Size 0805
  PTC P1K0.0805.2P.A PT1000, Class A, Size 0805 all other R - Vishhay CRCW08051K00FKEA,
  Standard Thick Film Chip Resistor, Size 0805
- polarized C - AVX TAJA105M020R
  Chip capacitor, Tantalum 20 V, Size A (EIA Code 3216)
  not polarized C - AVX 08055C104J4T2A
  Chip capacitor, Ceramic 50V , size 0805
- Active Components - Linear Technology Dual OpAmp LTC6081IMS8, and Analog Devices AD8422ARZ, both Package SOIC-8 3.9 x 4.9mm

- **PCB-Design**

The main steps for creating the schematic are provided below:

1. I created a hierarchical design by opening a main drawing and creating a 3 separate blocks for each Sheet.
2. Then in the sheets, I implemented schematic diagrams of the function Blocks to process between signal names. Use hierarchical labels Used for off-slide signals.Next, we connect the sheets on the top drawing.
3. Then, we run the Electronic Rule Check (ERC), which shows the errors and warnings in the schematic such as unconnected pins and connected output pins.
4. Lastly, we deal with the errors and make sure they are all eliminated.
   After that, we “Assign PCB footprints to the schematic symbols” and connect symbol and an appropriate package.

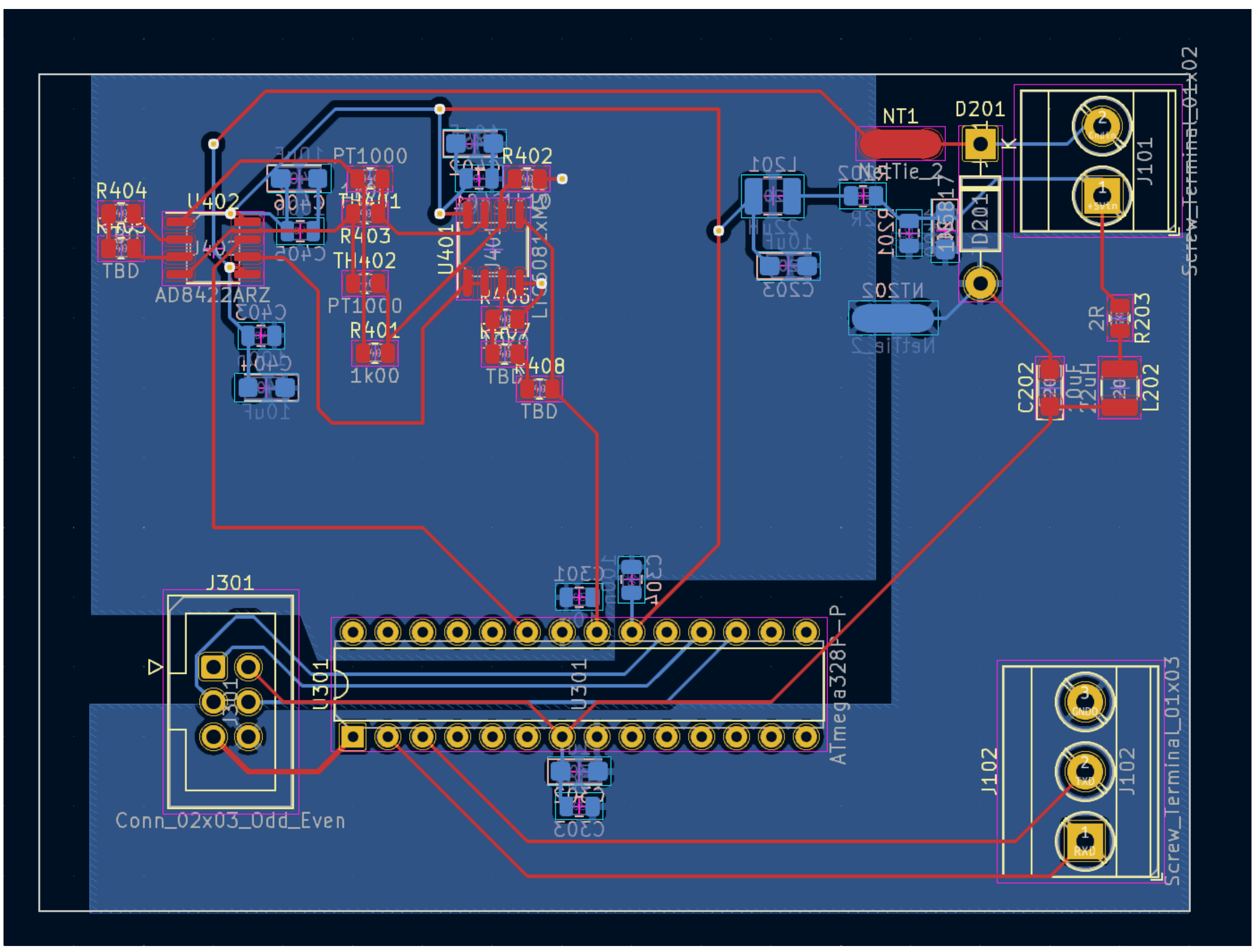
- **Routing**

1. Open the PCB editor and use the Board Setup feature on the File menu to remove all unused layers from the PCB. Under Design Rules / Predefined Sizes, define some tracks at 8, 16, 32, 60, and 80 mils.
2. Draw a given mechanical shape / dimension of the circuit board.

3. Then import the netlist from the schema editor. All components It looks like they are stacked somewhere and connected by air wires.
4. I then place the connectors and contacts. Using the defined functional blocks as guide, we place the components. I make sure to keep the analog and digital sections separate.
5. Next, I make a final check of the mechanical layout and see if all the device packages are okay.
6. At this point I should have a PCB with all components placed. All pins should be connected via air wires. Once I make sure that the air wires are as short and ‘unscrambled’ as possible, I can route the board.

- **Execution**

The final routed PCB design after the above steps were followed is :



The circuit is generally made up of the power, analog and digital parts. On the top right is the power block that powers the analog and digital blocks. It has the RLC filter for noise reduction in the voltage supplied.
The analog block(top left) is made up of temperature sensors within a Wheatstone bridge to detect changes in temperature, and the amplifiers amplify the signals within the analog block for detection at the digital block.

In the digital block(bottom center), the ADC conversion is done and values are stored.

- **Conclusion**

The efficiency of the device is heavily reliant on taking numerous precautions and adhering to various rules during the PCB design process. These precautions must be within the constraints set forth in the specification sheet, as per the customer's request, while considering important factors such as project timeline, costs, power consumption, and circuit function. To achieve desired performance, appropriate circuits must be developed, components must be chosen, and extensive testing must be conducted. The circuit schematics must then be created and used to place the components on the PCB. After routing, the PCB must be produced and tested, and any necessary modifications must be made. Ultimately, the final device can be delivered.

Throughout the project, the primary obstacles included familiarizing oneself with the software and navigating the smaller grid, which made schematic development initially challenging. Moreover, routing for optimal efficiency often required component repositioning, which occasionally led to broken connections. Additionally, warnings arose due to minor issues such as short stray traces, which necessitated further changes to ensure effective routing.

- **References**

1. Uwe Pagel & Prof. Dr. Ing. Werner Henkel , http://uwp-raspi-lab.jacobs.jacobs-university.de/03.0.specarealab/co-527-manual.pdf