Constructor University Bremen

CO-527-A

PCB Design and Measurement Automation

Spring 2024

Lab Report - Printed Circuit Board Design

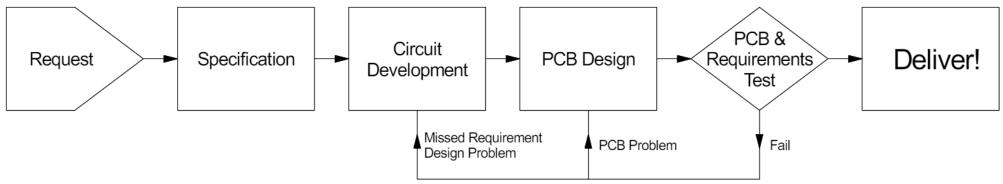
Abhigyan Deep Barnwal

Contents

Introduction

This lab offers an overview of electronic devices, exploring the various steps and aspects involved in transforming a circuit concept into a functional device.

General Design Flow



Step 1 - Request

The design process begins with a request from the customer, which includes a comprehensive specification outlining the complete requirements for the desired product. This written request is part of the contract between the customer and the developer. An incomplete, incorrect, or faulty product specification can lead to wasted money, project delays, or outright failure.

Step 2 - Specification

The product specification refers to a comprehensive and precise description of all requirements and information needed to develop the product according to the customer's demands. It is crucial that the specification is detailed and accurate to enable the developer to plan and prepare for efficient production.

Step 3 - Circuit Development

The development process for a circuit is initiated based on the request and specifications provided. The first step involves defining, developing, and simulating the circuits. Based on the requirements outlined in the specifications, the necessary components are then specified.

Step 4 - PCB Design

After completing the schematics, the mechanical setups are determined and finalized. The final step involves designing the PCB layout to meet the specified requirements.

Step 5 - PCB & Requirements test

Before the final design is delivered, the device undergoes a final testing process to ensure it functions correctly according to the requirements. During this step, any missed requirements, design problems, or PCB issues are identified and addressed, and necessary changes are applied to resolve any errors.

Step 6 - Delivery

After the PCB design is final, and all errors have been fixed, the design can then be delivered to the printer. It will then be manufactured and supplied to the customer.

That sums up the design flow, so we now move on to the lab, which involves getting a request, doing all the intermediate steps, and delivering a finished design.

Designing a PCB

Part 1 - Request and Specification

The request and specification was provided to us in the lab manual, simulating the experience of getting a requirement sheet for a product in a company. The given required specifications were as follows:

Function

Develop a circuit which measures temperature and transmits the result via RS232. Temperature range should be from −10º to +50ºC.

Accuracy

±(5%rdg + 4dgt)

Sample rate, stored history

2Val/min, Values of last two hours

Power

use a 5V ± 10% supply. Maximum power consumption 200mW.

Mechanical

usage inside dry rooms, maximal board size 61 X 48mm.

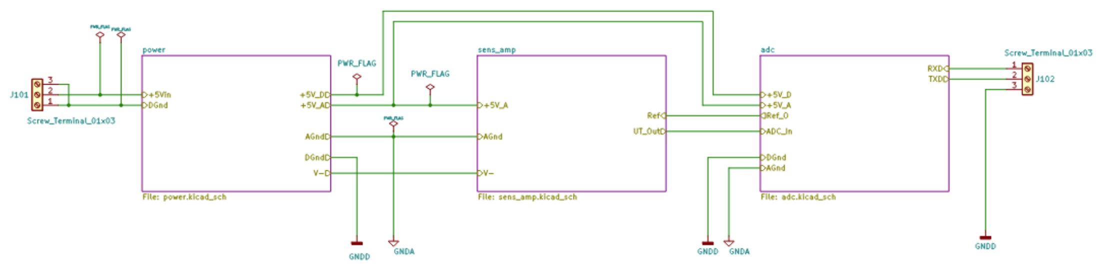
Time frame

6 weeks, as cheap as possible!!

Part 2 - Circuit Development

For our circuit, we split it into 3 parts: power, digital and analog.

This schematic connecting the three parts is shown below:

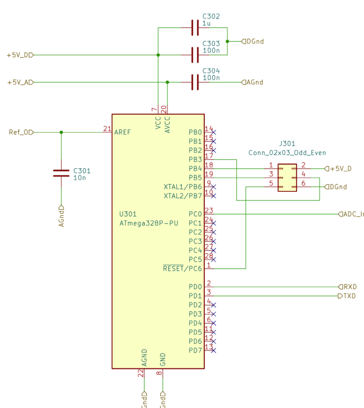


<u>Digital part</u>

For the digital components, including the ADC (Analog to Digital Converter), Timer, serial interface, and data memory, a single microcontroller is used that integrates all necessary elements. In this case, the Atmega 328-P processor on the Arduino board is utilized.

- Timer for measuring interval - available, almost any timing possible.
- Analog digital converter - an easy controllable 10 bit ADC is included. One sample every 30s can be easily measured.
- 1024 Byte RAM memory is available - we need space for 2 Val/min for two hours. So 2 Val/min $*$ 60 min $*$ 2 h = 240 Vals. Since one value has 2 Bytes, 480 Bytes are needed. That leaves enough RAM for the program to use
- Serial RS232 interface - Since we need an accurate reference voltage for the sensor circuit, it is convenient that the microcontroller contains a 1.1 V reference voltage source that can be used for the ADC and also for the analog part (1.074 mV/bit).

<u>Possible realization for the digital part</u>



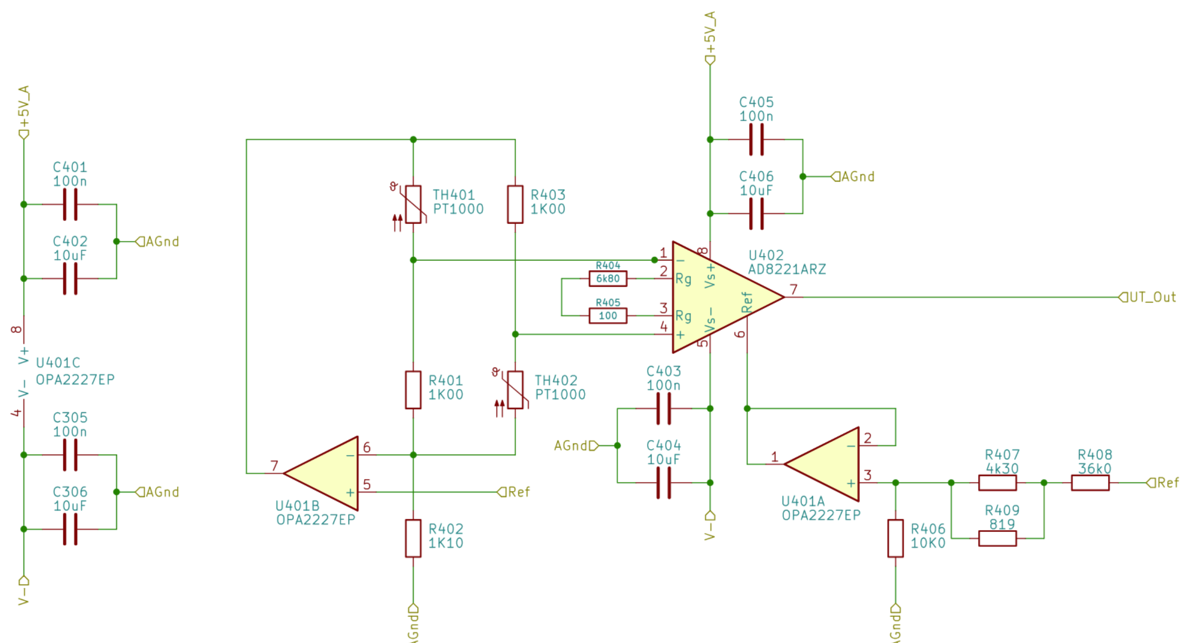
Making sure we have the above requirements, here is a possible schematic for the circuit. It's a very economical design, as we only need the microcontroller, the program connector, and blocking/filter capacitors for the reference and supply voltages.

Analog part

We need a reference voltage, a sensor circuit, and one or more amplifier circuits:

- The reference voltage is provided by the microcontroller. Since it is too weak, it needs amplification. One solution for this is to use a precision op-amp in a non-inverting configuration as a buffer.
- One or two PT1000 temperature sensors:
    - Inside a Wheatstone bridge to detect changes in resistance. Using a Wheatstone bridge is not simple in this case, as the output voltage should be as high and linear as possible over the entire range. A single PT1000 and a voltage source result in a low, non-linear output voltage. In the correct configuration, a second PT1000 doubles the output voltage. Supplying the Wheatstone bridge with a current source results in a linear response.
- The differential output of the bridge is amplified by an instrumentation amplifier, an integrated amplifier with differential input and configurable gain and offset.

Possible realization for the analog part



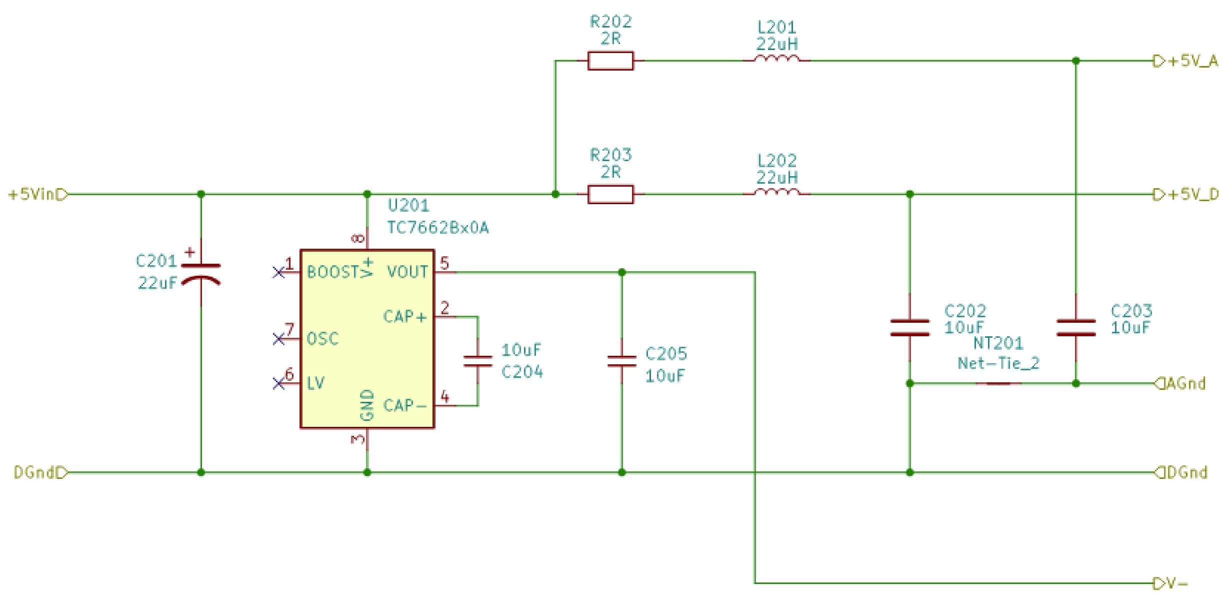
On the left, we see the Wheatstone bridge supplied by a constant current, and on the right, we see the instrumentation amplifier along with the reference buffer.

Power part

5V is the requested supply voltage of the circuit. After checking the different supply voltages of the required components we can see that we need a bipolar ± supply. The input voltage 5V can be used for the positive part, but a converter is required to generate a negative supply. To find a circuit we first need to determine the power requirements of all active components and the total power consumption.

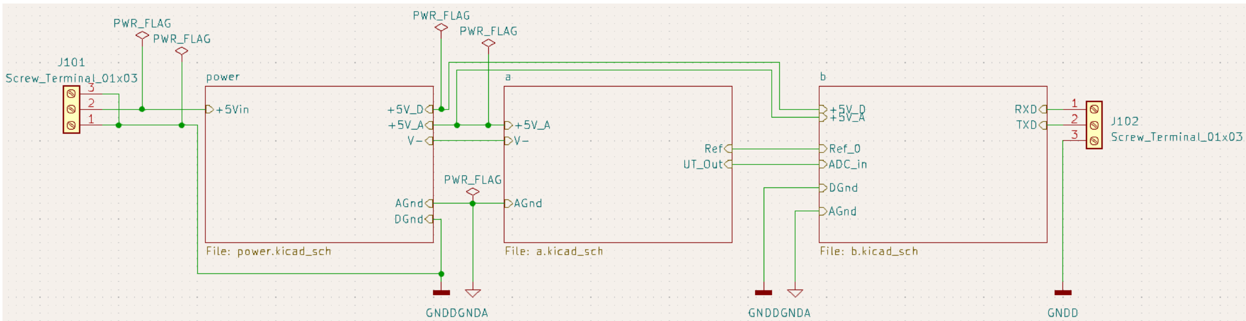
- The controller needs up to 10 mA in the worst case
- The amplifiers need up to 1 mA supply current
- The current for the Wheatstone reference is about 1 mA
- All this information comes from the data sheets of the components. To be safe the external supply should deliver 5 V ± 10% and ≈ 20 mA. That also means that the requirement of less than 200mW of power consumption is fulfilled.

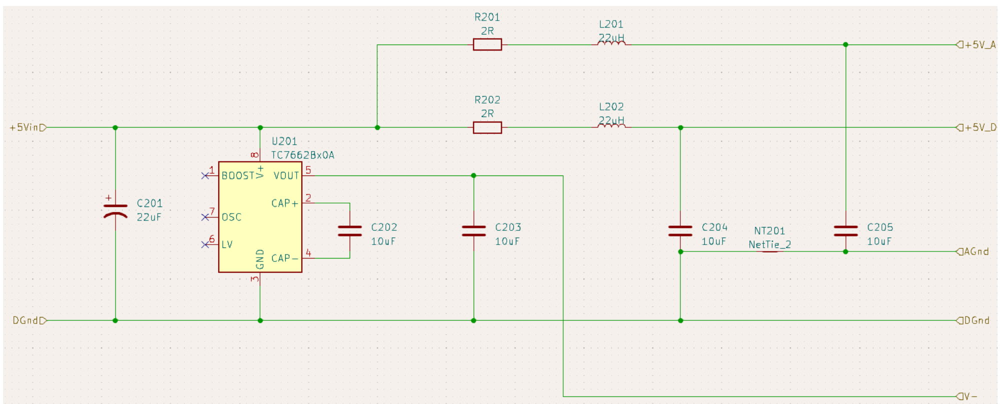
Possible realization for the power part

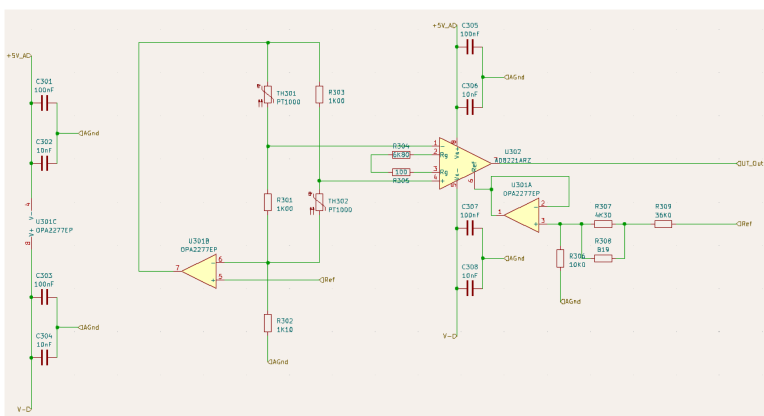


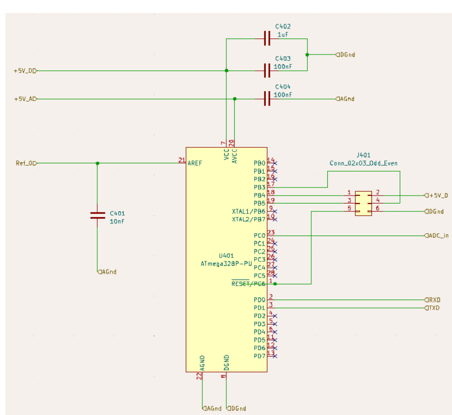
KiCad - Implementing the circuit

Now we have to implement the circuit designed above in KiCad:



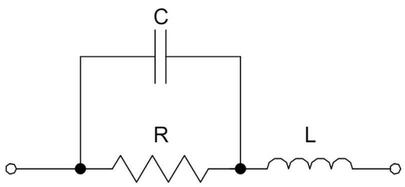


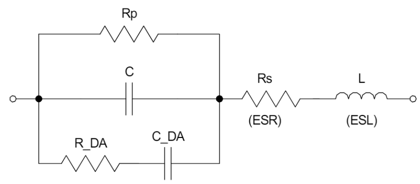


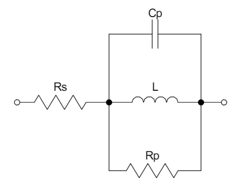


Choosing components

Now that we have drawn the circuit, we need to actually choose the real life components that we can use to implement the circuit. When doing this, we have to keep in mind the properties of the components that we use:







- RESISTORS: The model of a non-ideal resistor (above on the left) is given by a nominal resistance R bridged by a capacitor C_P which represents the capacitance between the end caps. L is the parasitic inductance of the leads. Typical values for a carbon-composition resistor with 1/4-inch leads might be 14 nH of series inductance and 1–2 pF of parallel capacitance. When dealing with resistors, we have to keep the following properties in mind: Tolerance, temperature coefficient, voltage sensitivity. power, thermoelectric effects, reactance, noise, failure mechanism
- CAPACITORS: The model of a non-ideal capacitor (above in the center) is given by the nominal capacitance C, shunted by a resistance R_P which represents insulation resistance or leakage. R_S is the equivalent series resistance (ESR) in series with the capacitor, representing the resistance of the capacitor leads and plates. The Inductance L is the equivalent series inductance (ESL), modeling the inductance of the leads and plates. When dealing with capacitors, we have to keep the following properties in mind: Tolerance, temperature coefficient, lossiness, insulation resistance or time constant, leakage current, dielectric absorption, self resonance
- INDUCTORS: The model of a non-ideal inductor (above on the right) is given by an ideal inductor L, the series resistance R_S, which arises from the winding wire and terminations, increasing with temperature. The parallel resistance R_P is due to magnetic core losses and varies with frequency, temperature, and current. The self-capacitance C_P of the winding is determined by the construction method of the component. When dealing with inductors, we have to keep the following properties in mind: Current carrying and saturation characteristics.

Given the above properties, we choose the following components:

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 Atmega programming adapters
  Connector J101
  J102 - Screw terminal 2 Pin, Grid 2.54

Power Part:

- 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

Analog Part:

- Bridge R - Metal Film Vishhay Y16241K00000T9R
  Ultra High Precision Foil Wraparound Chip Resistor,
  Tolerance 0.01%, TCR $\pm 1.8\,\mathrm{ppm}/°C$, Size 0805
  PTC P1K0.0805.2P.A PT1000, Class A, Size 0805
  all other R - Vishhay CRCW0805xxxxFHEAP Datasheet ,
  Semi-Precision Thick Film Chip Resistors, Tolerance 1%,
  TCR $\pm 50\,\mathrm{ppm}/°C$, 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 OPA2227EP
  Instr. amplifier AD8221ARZ
  both Package SOIC-8 3.9 x 4.9mm

KiCad - Implementing the chosen components

```
 1     C201 -            22uF : Capacitor_Tantalum_SMD:CP_EIA-7343-15_Kemet-W
 2     C202 -            10uF : Capacitor_SMD:C_0805_2012Metric
 3     C203 -            10uF : Capacitor_SMD:C_0805_2012Metric
 4     C204 -            10uF : Capacitor_SMD:C_0805_2012Metric
 5     C205 -            10uF : Capacitor_SMD:C_0805_2012Metric
 6     C301 -           100nF : Capacitor_SMD:C_0805_2012Metric
 7     C302 -            10nF : Capacitor_SMD:C_0805_2012Metric
 8     C303 -           100nF : Capacitor_SMD:C_0805_2012Metric
 9     C304 -            10nF : Capacitor_SMD:C_0805_2012Metric
10     C305 -           100nF : Capacitor_SMD:C_0805_2012Metric
11     C306 -            10nF : Capacitor_SMD:C_0805_2012Metric
12     C307 -           100nF : Capacitor_SMD:C_0805_2012Metric
13     C308 -            10nF : Capacitor_SMD:C_0805_2012Metric
14     C401 -            10nF : Capacitor_SMD:C_0805_2012Metric
15     C402 -             1uF : Capacitor_SMD:C_0805_2012Metric
16     C403 -           100nF : Capacitor_SMD:C_0805_2012Metric
17     C404 -           100nF : Capacitor_SMD:C_0805_2012Metric
18     J101 - Screw_Terminal_01x03 : TerminalBlock_Phoenix:TerminalBlock_Phoenix_MPT-0,5-3-2.54_1x03_P2.54mm_Horizontal
19     J102 - Screw_Terminal_01x03 : TerminalBlock_Phoenix:TerminalBlock_Phoenix_MPT-0,5-3-2.54_1x03_P2.54mm_Horizontal
20     J401 - Conn_02x03_Odd_Even : Connector_PinHeader_2.54mm:PinHeader_2x03_P2.54mm_Vertical
21     L201 -            22uH : Inductor_SMD:L_1210_3225Metric
22     L202 -            22uH : Inductor_SMD:L_1210_3225Metric
23    NT201 -        NetTie_2 : NetTie:NetTie-2_SMD_Pad0.5mm
24     R201 -              2R : Resistor_SMD:R_0805_2012Metric
25     R202 -              2R : Resistor_SMD:R_0805_2012Metric
26     R301 -            1K00 : Resistor_SMD:R_0805_2012Metric
27     R302 -            1K10 : Resistor_SMD:R_0805_2012Metric
28     R303 -            1K00 : Resistor_SMD:R_0805_2012Metric
29     R304 -            6K80 : Resistor_SMD:R_0805_2012Metric
30     R305 -             100 : Resistor_SMD:R_0805_2012Metric
31     R306 -            10K0 : Resistor_SMD:R_0805_2012Metric
32     R307 -            4K30 : Resistor_SMD:R_0805_2012Metric
33     R308 -             819 : Resistor_SMD:R_0805_2012Metric
34     R309 -            36K0 : Resistor_SMD:R_0805_2012Metric
35    TH301 -          PT1000 : Resistor_SMD:R_0805_2012Metric
36    TH302 -          PT1000 : Resistor_SMD:R_0805_2012Metric
37     U201 -      TC7662Bx0A : Package_SO:SOIC-8_3.9x4.9mm_P1.27mm
38     U301 -       OPA2277EP : Package_SO:SOIC-8_3.9x4.9mm_P1.27mm
39     U302 -       AD8221ARZ : Package_SO:SOIC-8_3.9x4.9mm_P1.27mm
40     U401 -    ATmega328P-PU : Package_DIP:DIP-28_W7.62mm
```

Once we are done choosing and implementing the components, we first use the electrical rules check function in KiCad, to ensure no minor and obvious mistakes have been made. Next, we move on to PCB Design.

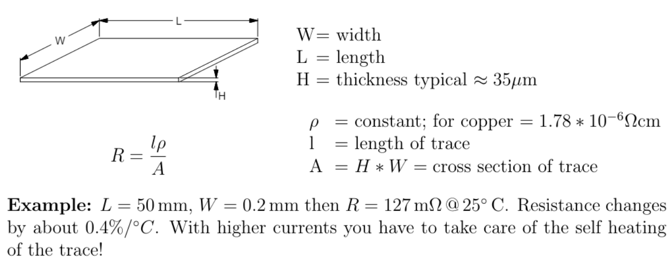
Part 3 - PCB Design

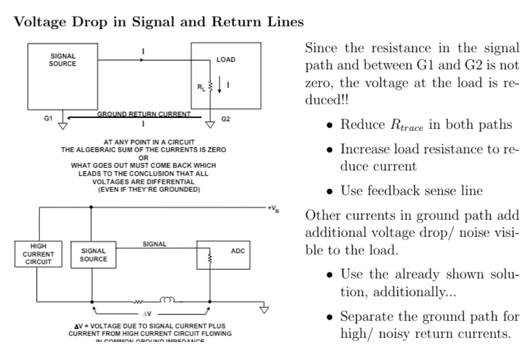
Once all the components have been assigned and have housings, we have to now do one of the most important parts of the PCB design process, which is routing. Other than lines in the schematic traces on the PCB and the PCB itself have physical properties. Depending on the circuit we have to take care about Ohm's Law, Kirchhoff's Law, electric, magnetic field theory, and transmission line theory. In general…

- Each trace on a PCB has a resistance and an inductance
- Together with another trace it forms a capacitor between two signals
- An unfavorable routed signal may form a single winding of a coil and behaves like a transformer adding voltage somewhere
- Depending on the circuit the impedance of a line has to fit to the rest of the system
- The ε_r of the PCB material and the mechanical dimension affects the value of the capacitance between traces
- The material of the PCB is not an ideal isolator. There might be leakage currents between traces and/ or layers
- The material cannot withstand any voltage

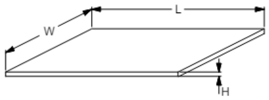
The goal is to keep all of these things in mind and minimize the unwanted effects. Going into more detail, here are some of the unwanted effects to specifically keep in mind:

- Resistance of traces





- Inductance of traces

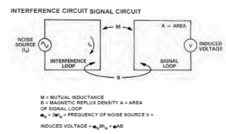


W= width
L = length
H = thickness typical $\approx 35\mu$m

$$\text{STRIP INDUCTANCE} = 2 * 10^{-7} L \left[\ln \frac{2L}{(W+H)} + 0.2235 \left(\frac{W+H}{L} \right) + 0.5 \right] H$$

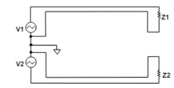
Example: $L = 10\,\text{mm}$, $W = 0.2\,\text{mm}$ then $L = 9.9\,\text{nH}$

The formula will given an approximate value, but is reasonably close to reality. At 10 MHz, an inductance of 0.9 nH has an impedance of 0.62Ω, and will induce $\approx 1\%$ error in a 50Ω system.



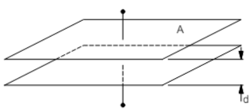
Mutual inductance is a magnetic coupling of coils, like in a transformer. In a circuit any KV loop forms a single winding of a coil. It receives and transmits magnetic fields. Voltage is induced in the receiving part.
The induced voltage is dependant on the current (strength of field) in the source, the distance of the source, and the area of the loop.

To avoid these effect keep the loop area small like in the example to the left. Make the distance between the loops as far as possible.



In general a ground plain can shield the magnetic field and also reduce the inductive effect of the traces.

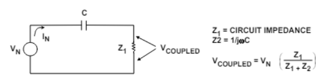
- Capacitance of traces



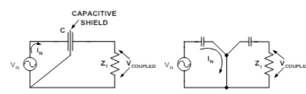
A = plate area
d = plate distance
ε_0 = permittivity $8.8542 * 10^{-12}$ F/m
ε_r = rel. permittivity ≈ 4.7 for FR4 material

$$C = \frac{\varepsilon_0 \, \varepsilon_r \, A}{d}$$

A 100 mm long trace with 0.25 mm width has ≈ 0.7 pF



Replacement circuit for two coupled traces. C is the stray capacitance, V_{in} is a source, and Z_1 is the sink. The voltage divider formula gives the noise voltage.



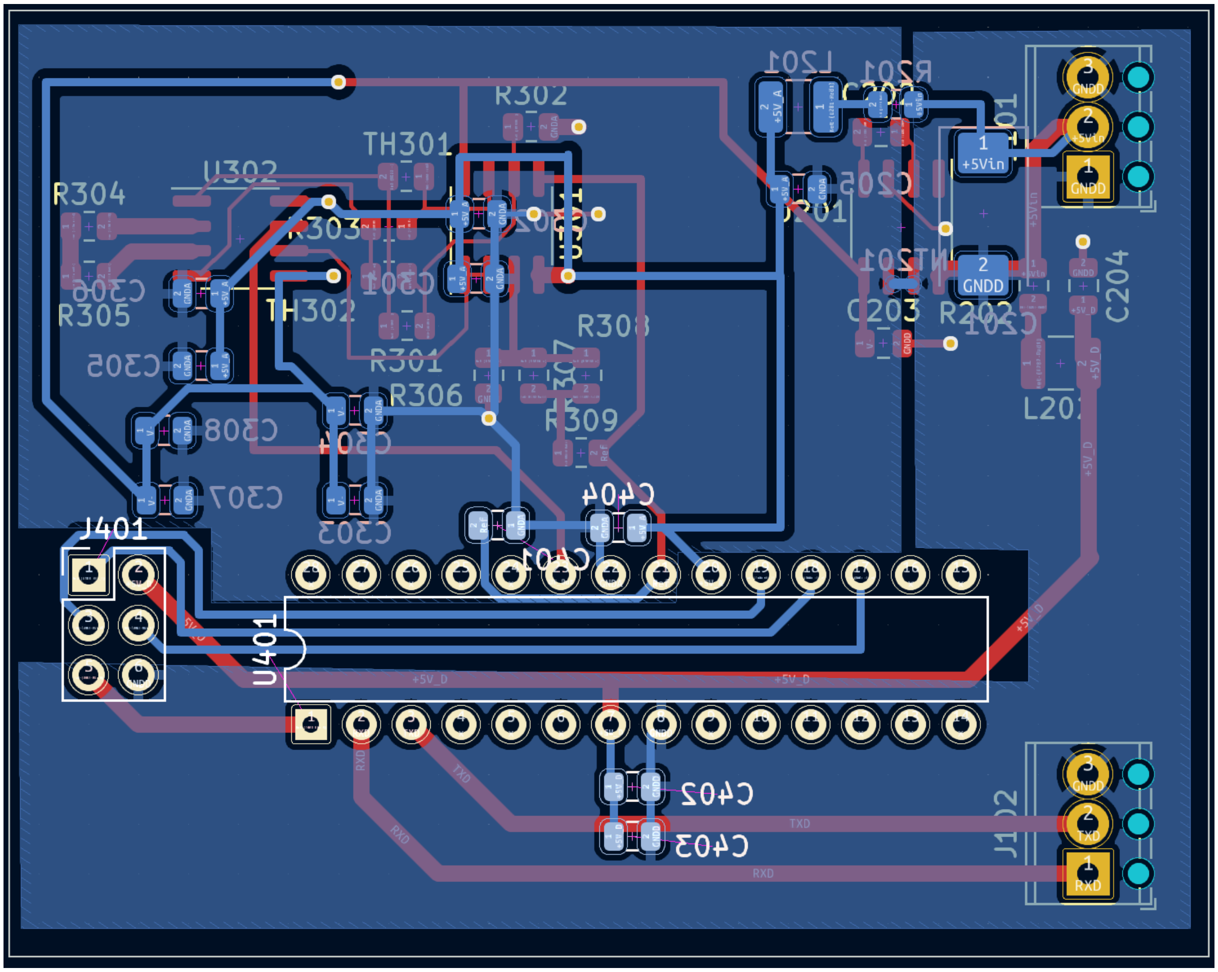
A ground plane ('Faraday shield') inside the PCB will eliminate the effect. The noise is shorted against ground in this case.

- Other mechanical of electrical requirements

Mechanical requirements from the board manufacturer must be met, such as minimum distances between copper surfaces, track widths, and drill hole sizes, typically detailed in a manufacturer's data sheet. Large components require proper fastening for mechanical stability, potentially needing screws or glue with adequate space provided. When connecting a conductor track to a solder point, the size ratio must be appropriate to avoid manufacturing and soldering issues. Dielectric strength must be considered, ensuring sufficient distance between live areas. Solder pads must fit element pads, with appropriately sized pads for SMD parts and round pads with drill holes to ensure reliable soldering.

<u>KiCad - Implementing the routing and final PCB Design</u>

The routing was done keeping all of the properties in mind. Afterwards, the Design Rules Checker was used, to make sure there were no obvious errors.



Conclusion

The lab module on PCB design aimed to provide a foundational understanding of the printed circuit board design process. It introduced various design rules and their implementation using KiCad software. The PCB was divided into power, analog, and digital sections, with each block designed separately while adhering to the design rules. After setting up the schematics, defining component packages, and generating a netlist, the mechanical setup, component placement, and signal routing were performed. Throughout the process, the properties of the circuit board were studied to ensure the design accounted for the components' characteristics. Key principles such as Ohm's Law, Kirchhoff's Law, electric and magnetic field theory, and transmission line theory were considered. Ultimately, the objective of the lab module was to introduce students to the design flow of an electronic device by guiding them through the creation of their first PCB.

References

Pagel, Uwe. *CO-527-A PCB Design Lab Manual*. 2024.

https://uwp-cu-lab.my-board.org/