CO-527-A

PCB Design and Measurement Automation

Spring Semester 2024

Course PCB Design and Measurement Automation – CO-527-A

Instructors	- Uwe Pagel, Res.I Room 37	Tel.: +49 421 200 3114
	- upagel (at) constructor.university	
	- Prof. Dr.Ing. Werner Henkel	Tel.: +49 421 200 3157
	- whenkel (at) constructor.university	
	- Prof. Dr. Mojtaba Joodaki	Tel.: +49 421 200-3215
	- m.joodaki (at) constructor.university	
Website	- http://uwp-raspi-lab.jacobs.jacobs-university.de/	

February 2, 2024

Contents

Part I

General remarks on the course

1. Lab Guidelines

1.1 Grading Scheme

1. The grade of this module is part of the overall grade of the course.
2. This module is graded by means of a report.

1.2 Attendance

Participation is highly recommended!! If you miss a session you have to develop the missed topics on your own. There will be no makeup for a session!!

1.3 Lab Report

The report is intended to summarize and demonstrate the design flow for an example circuit. The content should be:

1. Give an overview on the design flow.
2. Develop the specification sheet of the example project.
3. Introduce the given circuit and describe the general function.
4. Explain the selection of the components.
5. Use KiCad to draw the schematic of the circuit. Use a hierarchical design!
6. Give an overview about PCB properties and the design process. Summarize and explain the basic design rules.
7. Use KiCad to create a PCB.
8. Archive the complete KiCad project using 'File-Archive Project...' from the KiCad control program and deliver the archive together with the report.

Submit the whole material, i.e. report and KiCad project by email as usual.

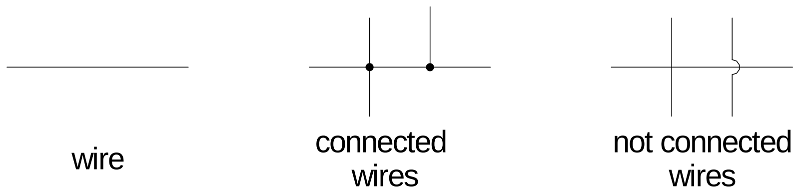
2. Manual Guideline

The manual and the course web-site contains all necessary information around the course.

2.1 Circuit Diagrams

Next is an overview about the used symbols in circuit diagrams.

Connections

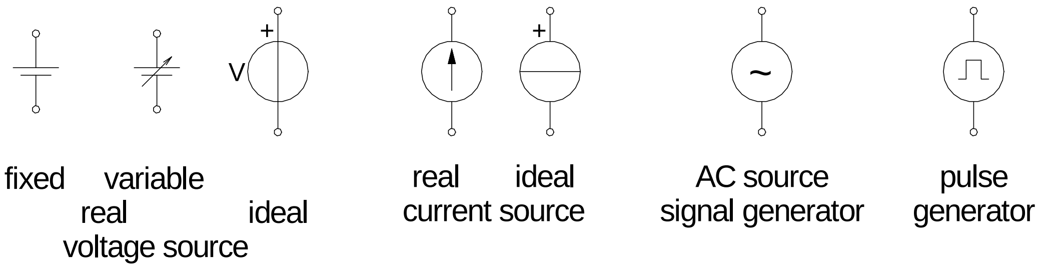


Connection are usually made using 1 or 0.5m flexible lab wires to connect the setup to an instrument or voltage source and short solid copper wires one the breadboard. In most of our experiments we consider these connections as ideal, i.e. a wire is a real short with no 'Impedance'. In the following semesters you will see that this is not true.

Instruments

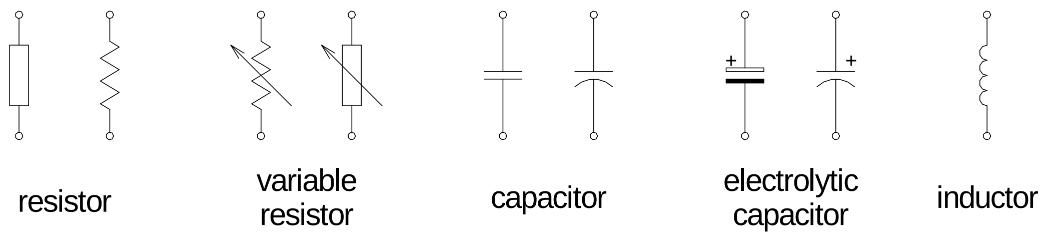
Since we have 'Multimeters' this symbol tells you how to connect and configure the instrument. Take care of the polarity. Be careful, in worst case you blow it!!!

Voltage/Current Sources



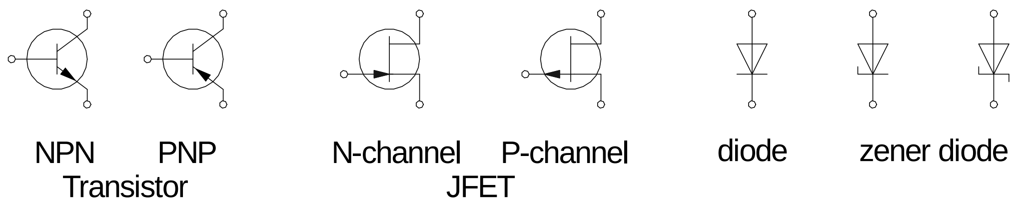
These are the symbols used in the manual. If you check the web and look into different books there are also other symbols in use!

Lumped Circuit Elements



There is a different symbol for every lumped circuit element. Depending which standard is used (DIN or IEC).

Semiconductors



Same as with the symbols before you may find different representations for every component!

2.2 Values in Circuit Diagrams

As you will see in the lab, we use resistors with colored rings. These rings represent numbers or a multiplier. Most of the resistors have five rings. Three digits for the value, one multiplier for the dimension, and one for the tolerance. In the circuit diagrams we have a similar scheme. There are three digits and a dimension. The letter of the dimension also acts as the comma i.e.:

1R00, 10R0, 100R	for	$1\,\Omega$, $10\,\Omega$, $100\,\Omega$	$(= Value * 10^0)$
1K20, 10K0, 100K	for	$1.2\,\mathrm{K}\Omega$, $10\,\mathrm{K}\Omega$, $100\,\mathrm{K}\Omega$	$(= Value * 10^3)$
1M00, 10M0	for	$1\,\mathrm{M}\Omega$, $10\,\mathrm{M}\Omega$	$(= Value * 10^6)$

The numbering for capacitors in the circuit diagram is similar. Only the dimension differs. Instead R, K, M (Ω, $\mathrm{K}\Omega$, $\mathrm{M}\Omega$) we have μ, n, or p (μF, nF, pF) (i.e. 1n5 means 1.5nF). The value is printed as number on the component.

Part II

Experiments

3. Introduction

3.1 Objective

This lab is intended to give an insight into the design process of an electronic device. In the following we will talk about the different steps and some aspects that have to be considered when converting a circuit idea into a real device. The whole process will be demonstrated with an example. The focus will be on the basics for creating a printed circuit board.

3.2 Prerequisites

All needed extra information like this handout, books and articles about the topic, data sheets, and examples are available on the web site

http://uwp-raspi-lab.jacobs.jacobs-university.de/

As prerequisite download and install the EDA software 'KiCad' from

http://www.kicad.org

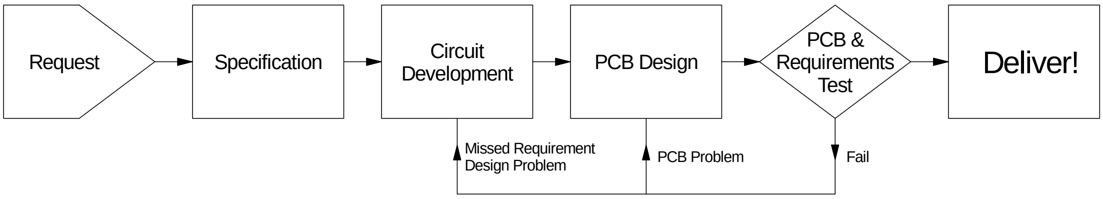
'KiCad' is a **'A Cross Platform and Open Source Electronics Design Automation Suite'**. That means it is an software tool for the creation of electronic schematic diagrams and PCBs artwork. We use this to design a small circuit board.

On the website you will find additional literature from Analog Devices and Texas Instruments on circuit design. This also discusses PCB design. A book that covers a complete design is

The Circuit Designer's Companion - Author Peter Wilson

4. General Design flow

Below is a diagram of the general design flow:



1. The design starts with an 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.

4.1 Request

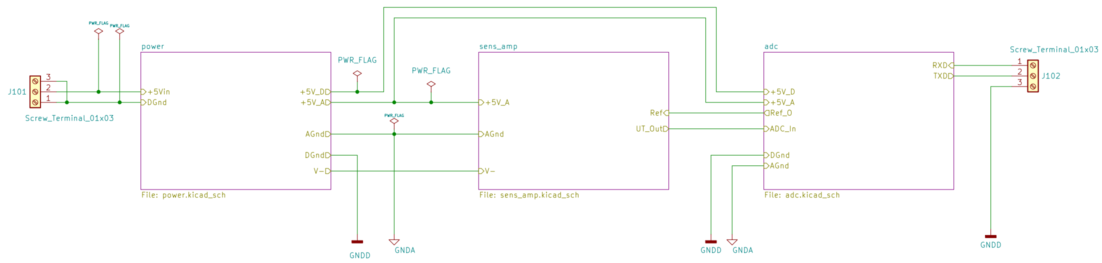
The complete requirements should be collected in the so-called specification and are part of the written contract between the customer and the developer. An incomplete, incorrect or faulty product specification can have a number of consequences, e.g. wasted money, delay or, in the worst case, the failure of the entire project.
To demonstrate the design flow we start a simple example project. The following information/ requirements has to be included into the specification sheet:

- Function:
  Develop a circuit which measures temperature and transmits the result via RS232. Temperature range should be from $-10°$ to $+50°C$.
- Accuracy: $\pm$(5%rdg + 4dgt)
- Sample rate, stored history: 2Val/min, Values of last two hours
- Power: use a $5V \pm 10\%$ supply. Maximum power consumption 200mW.
- Mechanical: usage inside dry rooms, maximal board size 61 X 48mm.
- Time frame: 6 week, as cheap as possible!!

5. Specification

5.0.1 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

5.0.2 Specification of the digital part.

What is needed and what might be used?
An ADC (Analog to Digital Converter), a Timer, a serial interface, and memory for data is needed. A single micro controller contains all these components, e.g. an Atmega168P processor like used on the Arduino board.

Check of the requirements (look into data sheet!!):

- Timer for measuring interval - available, almost any timing possible.
- Analog digital converter - an easy controllable 10 bit ADC is included. One sample every 30s is no problem.
- 1024 Byte RAM memory is available, needed is space for 2 Val/min for two hours. So

  $$2\,\text{Val/min} * 60\,\text{min} * 2\,\text{h} = 240\,\text{Vals}$$

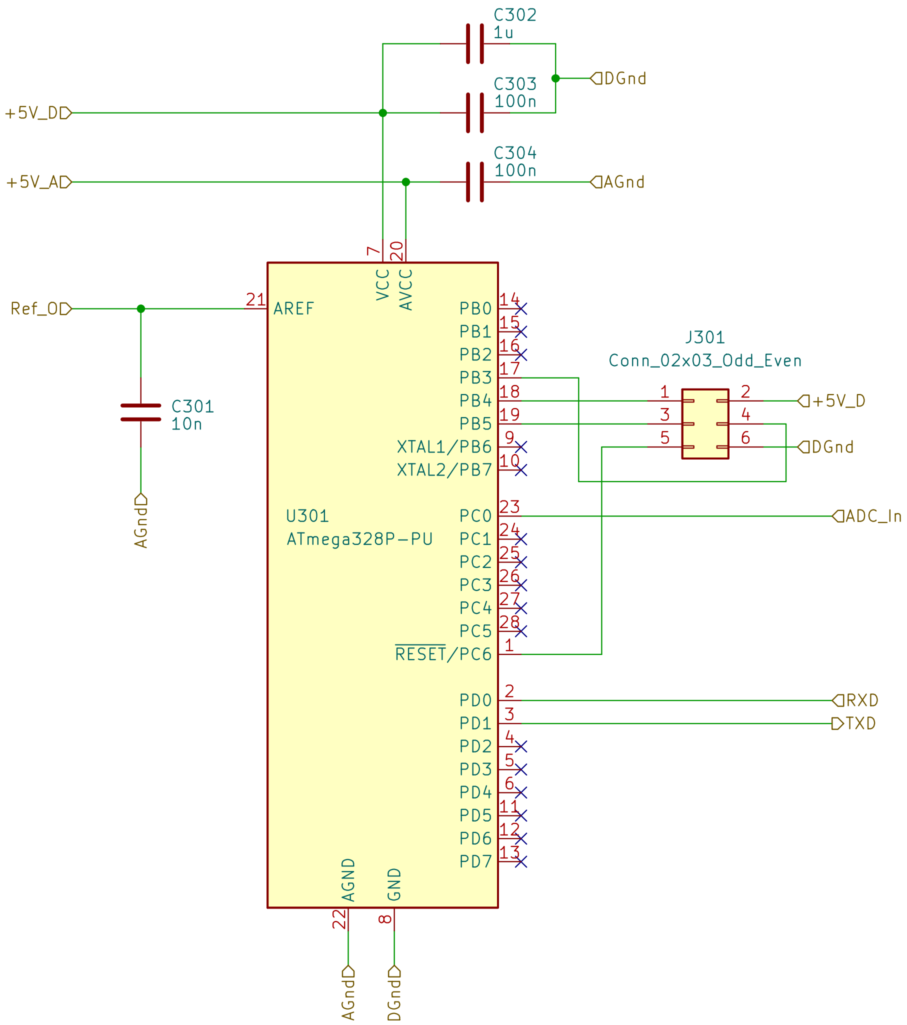
  Since one value has 2 Byte 480 Bytes are needed. That leaves enough RAM for program use!
- Serial RS232 interface - available.

... and as bonus on top ...

Since we need an accurate reference voltage for the sensor circuit, it is convenient that the MC 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).

Possible realization of digital part

It's a very 'economic' design!! We need only the micro controller, the program connector and blocking/ filter capacitors for the reference and supply voltages.



5.0.3 Specification of Analog

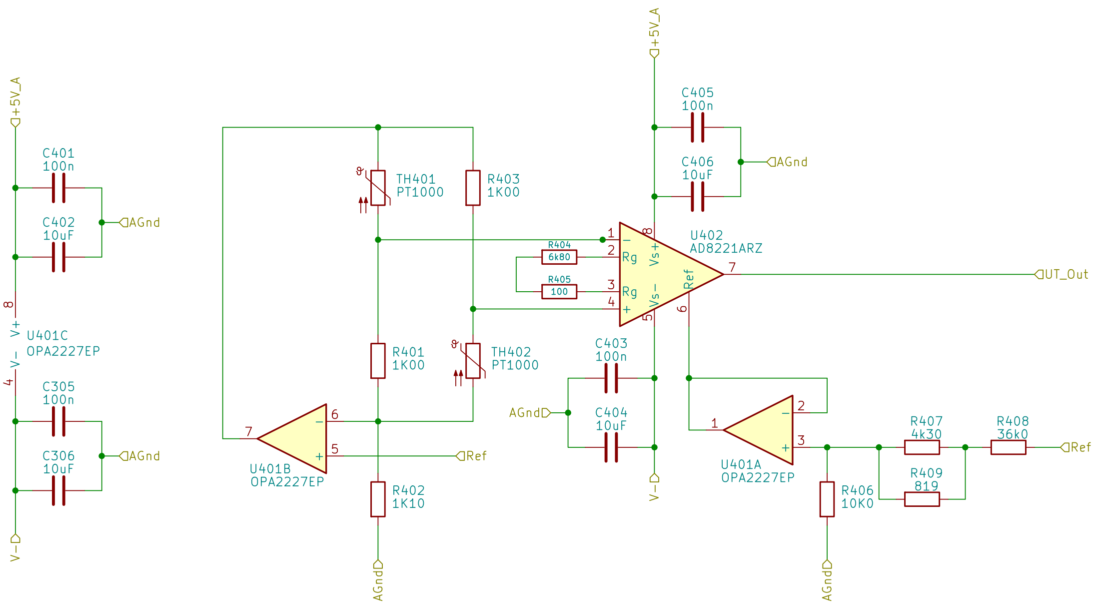
What is needed and what might be used?

A reference Voltage, a sensor circuit, one or more amplifier circuits

- The reference voltage is provided by the MC. Check the data sheet! Since it is to weak it needs amplification. 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. The usage of a Wheatstone Bridge is not so simple in this case! The output voltage should be as high as possible and linear over the entire range. A single PT1000 and the use of a voltage source results 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. A detailed description is given in a document on the course web site. One OpAmp two PT1000s and two resistors are requires to create the circuit.
- The differential output of the bridge is amplified by an instrumentation amplifier. That is an integrated amplifier with differential input and configurable gain and offset.

Possible realization of the analog part

There are several possible solutions. Here is simple solution. A short explanation is given in the 'Design, Engineering Part calculations' section.



To the left there is a Wheatstone bride supplied by a constant current. To the right is the instrumentation amplifier together with the buffer for the reference.

5.0.4 Specification of Power

What is needed and what might be used?

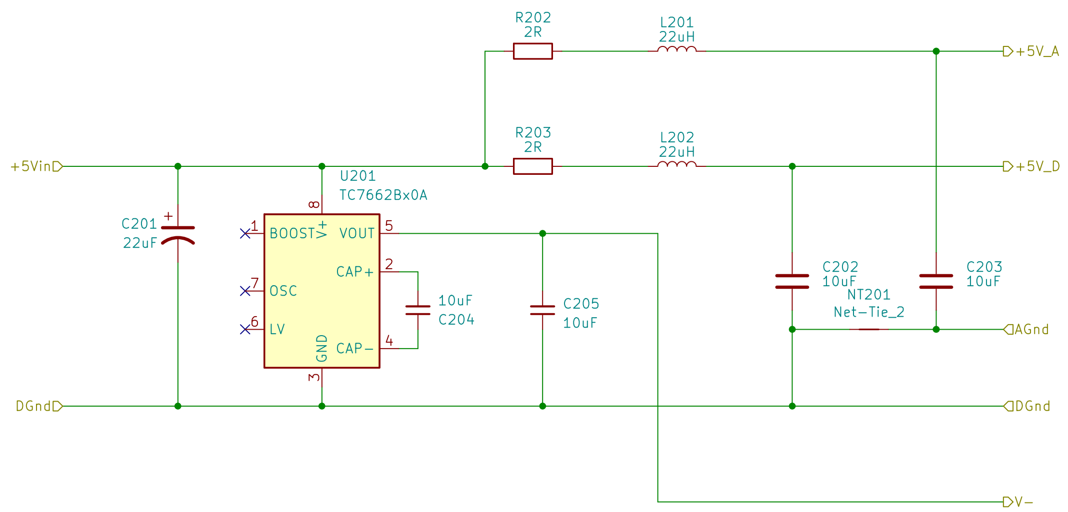
5 V is the requested supply voltage of the circuit. After checking the different supply voltages of the required components we can see that we need a bipolar $\pm$ supply! The input voltage 5 V can be used for the positive part, but a converter is required to generate a negative supply.
To find a circuit we first need to determine the power requirements of all active components and the total power consumption! For each functional block, we need to estimate the current and then add it up.

- The controller needs up to 10 mA in worst case (data sheet).
- 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 save the external supply should deliver 5 V $\pm$ 10% and $\approx$ 20 mA. That means also the requirement of less them 200mW is fulfilled.

Possible realization of the power part

Since the supply should have 5 V and all components can use 5 V we don't need an extra regulator for the positive supply. For the negative supply we use a charge pump voltage converter. It transfers a positive voltage to a negative one with nearly the same value. Additional we need LC filters to reduce ripple/noise from the power source!!



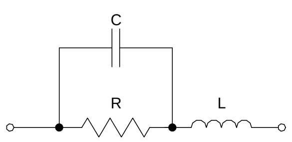
The collection of circuits to be used is complete now. Next step is to think of the needed components and their properties.

6. Component properties

In the laboratory, we have not yet focused on all the properties of the components used. We have considered them as ideal. For a real design, however, the components must be selected according to their specifications. The stability and accuracy depends on the design and material of the components. For the simulation of a "real" circuit, a model of the real component with all interference elements should be used. We must therefore first make an excursus to the properties of real components!!!

6.1 Resistors

A model of a non-ideal resistor (carbon-composition)



The nominal resistance R is bridged by a capacitor C_p, which is the capacity between the end caps. L is the parasitic inductance of the leads. Typical values for the carbon-composition resistor (with 1/4-inch leads) might be 14 nH of series inductance and $1-2$ pF parallel capacitance.

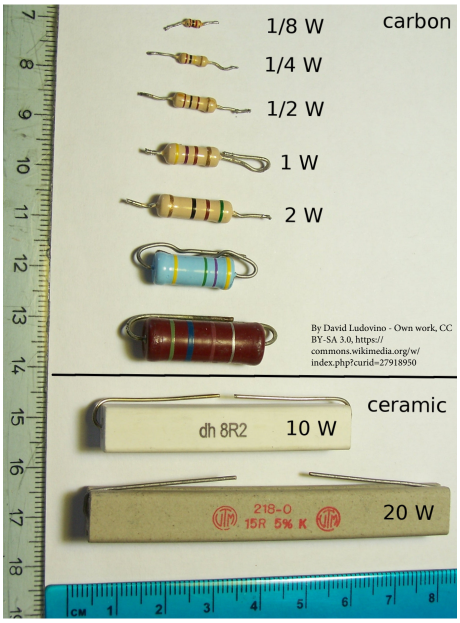
6.1.1 Resistor Properties

- Tolerance - Difference from nominal value
- Temperature Coefficient - The resistor value changes with temperature, the coefficient is given in $ppm/^\circ$C
- Voltage sensitivity - A resistor can only withstand a limited voltage. Dependent on the material the resistance changes. A voltage coefficient is given in $ppm/volt$.
- Power - a resistor can only withstand limited power. Of course power dissipation also affects the resistance (temperature coefficient)!
- Thermoelectric Effects - The connections between the resistive material and the leads form thermocouple junctions! Depending on the material the voltage is between $< 1\mu V/^\circ$C and $400\mu V/^\circ C$. The two(!) voltages usually cancel each other out as long as the two connections are kept at the same temperature.
- Reactance - the physical setup exhibits inductance and capacitance. These values are in the range of μH and pF. Of course wire wound resistors may add additional inductance, depending on how they are build.
- Noise - dependent on the resistive material and the frequency noise is induced.
- Failure Mechanism, Aging - Resistors change with age. A 'burn in' might be necessary to stabilize the value. Second, resistors behave different in case of failure. Depending on the type they turn into an open or a short circuit.

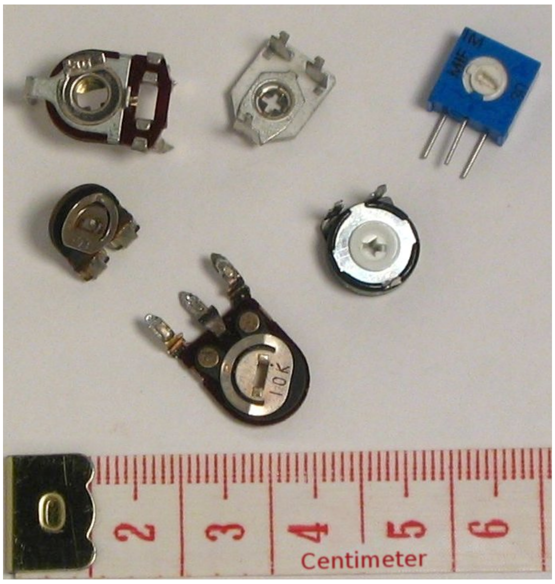
Types of resistors with main properties

Type	Ohmic Range	Power Range	Tolerance	Tempco Range in ppm/°C	Application
Carbon Film	$2.2 > 10$M	$0.25 > 2$W	5%	$-(150 > 1000)$	General purpose
Carbon Composite	$2.2 > 10$M	$0.25 > 1$W	10%	$+400 > -900$	Pulse, low inductance
Metal film (standard)	$1 > 10$M	$0.125 > 2.5$W	$1\%, 2\%, 5\%$	$\pm(50 > 200)$	General purpose
Metal film (high ohm)	$1 > 100$M	$0.5 > 1$W	5%	$\pm(200 > 300)$	High voltage
Metal glaze	$1 > 100$M	0.25W	$2\%, 5\%$	$\pm(100 > 300)$	Small size
Wire wound	$0.1 > 33$k $(0, 01..100)$M	$2 > 20$W ($10 > 100$W aluminum)	$5\%, 10\%$	$\pm(75 > 400)$	High power
Metal film (precision)	$5 > 1$M	$0.125 > 0.4$W	$0.05\% > 1\%$	$\pm(15 > 50)$	Precision
Wire wound (precision)	$1 > 1$M	$0.1 > 0.5$W	$0.015\% > 0.1\%$	$\pm(3 > 10)$	Extra precision
Bulk Metal (precision)	$1 > 200$k	$0.33 > 1$W	$0.005\% > 1\%$	$\pm(1 > 5)$	Extra precision
Resistor networks and arrays	$10 > 1$M	$0.125 > 0.3$W per element	2%	$\pm(100 > 300)$ ± 50 tracking	Multi resistor
SM chip film	$0 > 10$M	$0.1 > 0.5$W	$1\%, 2\%, 5\%$	$\pm(100 > 200)$	Surface mount

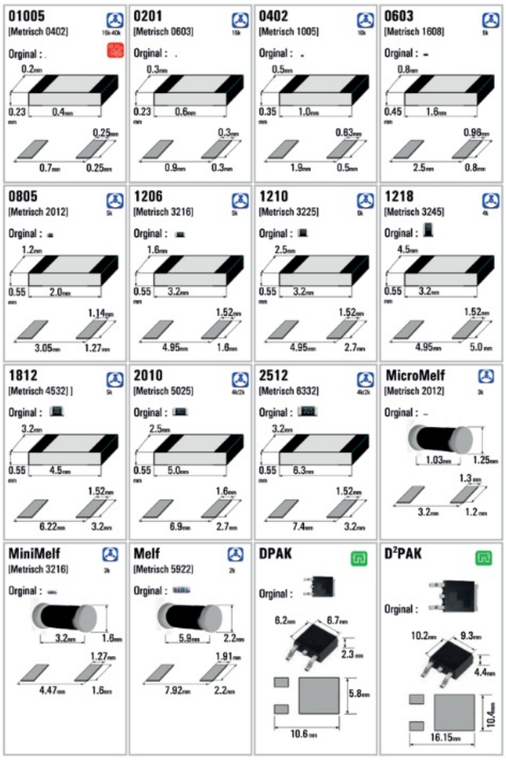
6.1.2 Types of ohmic resistors



Through Hole Technology



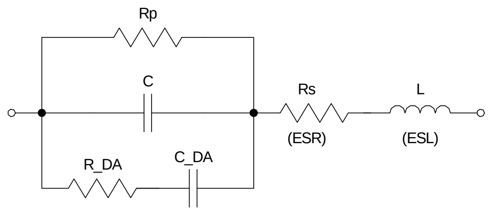
Variable resistors



Surface Mounted Device (SMD)

6.2 Capacitors

A model of a non-ideal capacitor



The nominal capacitance, C, is shunted by a resistance R_p, which represents insulation resistance or leakage. R_s is the equivalent series resistance, or ESR in series with the capacitor and represents the resistance of the capacitor leads and plates. Inductance L is the equivalent series inductance, or ESL, modeling the inductance of the leads and plates.

6.2.1 Types of capacitors

The divisions are best made by the dielectric. The following types are used for general purpose applications.

Film:	Polyester, Polycarbonate, Polypropylene, Polystyrene
Paper Ceramic:	Single layer, barrier layer, high-K, low-K
Multilayer:	COG, X5R, X7R, Z5U
Electrolytic:	Non solid and solid aluminum, solid tantalum

There are some other types for special applications.

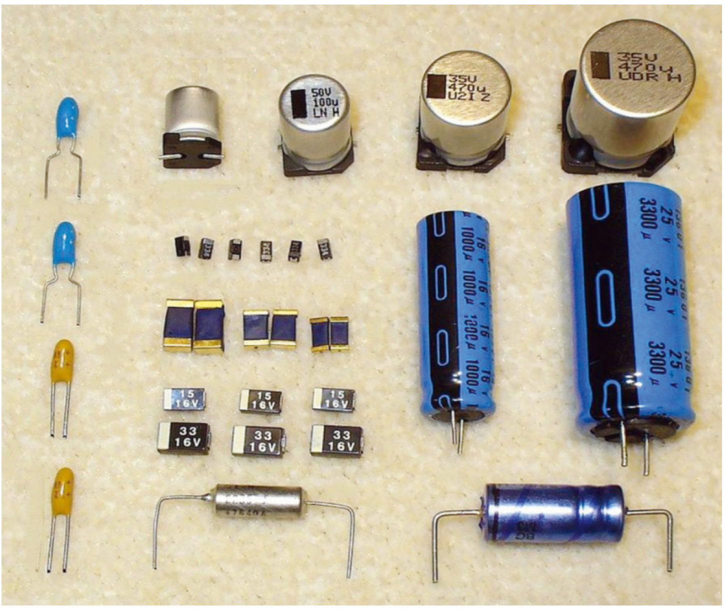
6.2.2 Capacitors Properties

- Tolerance - Difference from nominal value
- Temperature coefficient - the change in capacitance C with temperature. Given in ppm/°C or as a percentage change of C over the operating temperature range.
- $\tan\delta$ - Describes the lossiness of the component and is the ratio of the resistive and reactive parts of the impedance, R/X, where $X = \omega C$
- Insulation resistance or time constant - Measure of the DC leakage and may be quoted as the product $R_{dc} * C$ in seconds or as R_{dc} in MΩ.
- Leakage current - describes the same characteristic, more usually quoted for electrolytics.
- Dielectric absorption - Measure of the 'voltage memory' characteristic, which occurs because dielectric material does not polarize instantly but needs time to recover the full charge; it is defined as a percentage change in stored voltage a given time after the sample is taken.
- Self resonance - at high frequencies C, ESR, and ESL form a low-Q circuit.

Overview about different types of capacitors

Type	Capacitance Range	Typical DA	Voltage Range	Tolerance	Tempco	Applications
Mica	$< 1\mu$F	0.0003%	> 1000V	$>= 1\%$	$\pm(50 > 200)$ ppm/°C	HF, precision
Polyester	$1\,$n $> 15\mu$F	$0.3\% > 0.5\%$	$50 > 1500$V	5%, 10%, 20%	$\pm 5\%$ over $-55 > 100$°C	Coupling and decoupling
Polycarbonate	$100\,$p $> 15\mu$F	0.1%	$63 > 1000$V	5%, 10%, 20%	$< 1\%$ over $-55 > 100$°C	Low tc, timing and filtering
Polypropylene	$100\,$p $> 10\mu$F	$0.001\% > 0.02\%$	$63 - 2000$V	1%, 5%, 10%	$\pm 2\%$ over $-55 > 100$°C	High power, high frequency
Polystyrene	$10\,$p $> 47\,$nF	$0.001\% > 0.002\%$	$30 - 630$V	$1\% > 10\%$	-125 ppm/°C	close tolerance, low loss
Metalized paper	$1\,$n $> 0.47\mu$F	$0.003\% > 0.02\%$	$250\,\mathrm{V}_{AC}$	$\pm 20\%$		Mains RFI suppression
Ceramic single layer	$1\,$p $> 47n$F		$50 > 6\,$KV	$2\% > 20/80\%$	dep. on dielectric	general purpose and HV
COG / NPO	$1\,$p $> 27\,$nF	$< 0.1\%$	$50 > 200$V	2%, 5%, 10%	± 30 ppm/°C	Sensitive and timing
X7R	$1 > 680\,$nF		$50 > 200$V	5%, 10%, 20%	Nonlinear $\pm 15\%$ over $-55 > 125$°C	Coupling and decoupling
Y5V, Z5U	$1\,$n $> 10\mu$F		10V, 16V 50V, 100V	20% $-20\% + 80\%$	Nonlinear $+22\%, -56\%$	Coupling and decoupling
Electrolytics	$1 > 4700\mu$F		$6.3 > 100$V	$\pm 20\%$		general purpose
Aluminum oxide	$0.1 > 68000\mu$F	very high	up to 450V	$-10\% > 30\%$		Storage, decoupling
solid Aluminum	$0.1 > 68\mu$F	very high	$6.3 > 40$V	$\pm 20\%$		general purpose
Tantalum	$0.1 > 150\mu$F	very high	$6.3 > 35$V	$\pm 20\%$		general pupose, small size

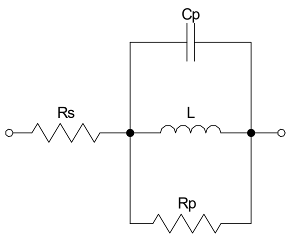
6.2.3 Types of capacitors



Selection of through hole and surface mounted C

6.3 Properties of Inductor

A model of a non-ideal inductor



- L is the 'ideal' inductance of the device
- R_S is the series resistance due to the winding wire and terminations, increasing with temperature
- R_p is an equivalent parallel resistance due to the magnetic core losses and is variable with frequency, temperature, and current
- C_p is the self-capacitance of the winding, determined by the method of construction of the component

R_S and R_p may be lumped together into an equivalent series resistance R_{eq}, in which case the overall lossiness of the component is quoted as its Q, which is given by $\omega L/R_{eq}$. The reciprocal of Q is $tan\delta$, which is analogous to the capacitor $tan\delta$.

Inductors are far less common as components than are resistors and capacitors. As for precision components, they are even more rare. This is because they are harder to manufacture, less stable, and less physically robust than resistors and capacitors. It is relatively easy to manufacture stable precision inductors with inductance from nH to tens or hundreds of μH, but larger valued devices tend to be less stable, and large.
Circuits are designed, where possible, to avoid the use of precision inductors, except in tuned circuits for high frequency narrow band applications. They are widely used in power filters, switching power supplies and other applications where lack of precision is unimportant.
The important features of an inductors are their current carrying and saturation characteristics, and their Q. If an inductor consists of a coil of wire with an air core, its inductance will be essentially unaffected by the current it is carrying. On the other hand, if it is wound on a core of a magnetic material (magnetic alloy or ferrite), its inductance will be non-linear, since at high currents, the core will start to saturate. The effects of such saturation will reduce the efficiency of the circuitry employing the inductor and is liable to increase noise and harmonic generation.
The Q (or 'Quality') factor of an inductor is the ratio of the reactive impedance to the resistance $Q = 2\pi fL/R$. R is the HF and not the DC resistance!!

7. The Engineering Part

The next step is to check the design. The properties required in the specification must be achieved. Various components must be dimensioned and the operating limits of the components and the required tolerances must be determined. The search for the components can be a lengthy process depending on the problem. It is necessary to read data sheets to find the components. The circuit must be simulated or built on a breadboard. The function must be checked and an error estimation over the operating range must be made. In the final step, the housings of the various components are then determined.
After simulation the following components might be used.

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 Atmega programming adapters
     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,
     Tolerance 0.01%, TCR $\pm 1.8\,\text{ppm}/°C$, 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 $\pm 50\,\text{ppm}/°C$, 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

Here all components are defined and all packages for the PCB are known. Last missing information is the mechanical demention of the PCB. Length x width should be 61 x 48mm (2400 x 1900 mil).
The circuit should be already drawn here. Now we can start with the PCB design!!

8. PCB-Design - General Introduction

8.1 PCB-Design using EDA

EDA means **E**lectronic **D**esign **A**utomation

- In the past every step was done on paper and breadboard. For small designs this is still sufficient!
- For complex designs it is easier and more efficient to use computers. In the simplest case you need one program for every step. E.g. design tools like VHDL, simulators like LTSPice, schematic capture and PCB design programs like KiCad.
- More convenient is to use an integrated solution. Every step is made inside one environment, this keeps the design consistent and well documented through all design steps. The final output is a printed circuit board with all information for the manufacturer and the testing facility.
- Complete integrated solutions are usually expensive. Here we use a combination. LTSpice and MatLab for simulation, and KiCad for schematic capture and PCB design.

8.2 Design Tool - KiCad - Overview

For this course I have chosen 'KiCad' as design tool. It is widely used, available for all major operating systems, and open source. It is under active development and the functionality increases.

8.2.1 Functions of the Package

- Schematic capture of circuit diagrams.
- Defining PCBs and placing components.
- Manual routing of the connections.
- Prepare the documents for manufacturing of the circuit board and assembling.

8.2.2 Components

- **Schematic editor**
  In the schematic editor you draw a circuit diagram. This is the functional description of the circuit. It shows the components and their connections. The component are selected from a library or have to be created. Connections are shown by lines. In the 'properties' of each symbol the footprint for the PCB must be defined. From the schematics a net list of the whole circuit is created.

- **PCB editor**
  The net list from the schematic will be imported to the PCB editor. The PCB editor is used to determine the mechanical properties of the board, the placing of the components, and the routing of the traces.

- **Library manager/ Symbol and Footprint editor**
  A device you want to use in a circuit always consists of a symbol, the representation in the schematic, and a package as the mechanical representation in the PCB editor. Symbol and package are connected via pin (symbol) and pad (package) descriptions. The libraries already contain predefined symbols and packages from many manufacturers. Not contained symbols and packages have to be defined on your own.

- **Gerber viewer**
  The Gerber viewer is used to check the production files generated by the PCB editor.

The installation of the program is described in the tutorial 'First steps in KiCad' on the course website. Get an overview of the program and edit the example. Use the different KiCad manuals and help files for a detailed description of the functions!

9. PCB-Design

9.1 Schematic capture

Before you can start to design the PCB you have to create the complete schematics. The steps for drawing the circuit are:

1. Create a hierarchical Design, i.e. open a top drawing and first draw a block for each function. The functional blocks are called 'Sheets' connected by 'Sheet Pins'. The module name should be descriptive. Each 'Sheet Pin' has to fit a signal name inside the module.

2. Move into the sheet and draw the schematic of the functional blocks. Take care of the signal names. Use hierarchical labels for signals used outside a sheet. In our case all needed components should be available in the supplied libraries.

3. Connect the sheets on the top drawing.

4. If you are done run the 'ERC' (Electrical Rule) check. This will give you errors and warnings like unconnected pins or connected output pins. Check each item carefully and decide if it is a real problem. You will find that you get also false messages, but there are always real errors included like unconnected pins.

5. Once all real errors have been eliminated, use **'Assign PCB footprints to schematic symbols'** and connect the symbols to the correct housings.

Until here the circuit is complete and we should have a set of schematics. All component should have reference identifiers and packages assigned. Next step is to define the mechanical setup of the circuit board, the placing of the components, and the routing of the signals. Before we can start the PCB-editor first we need to talk about the properties of a circuit board.

9.2 Routing - Theoretical Aspects

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.

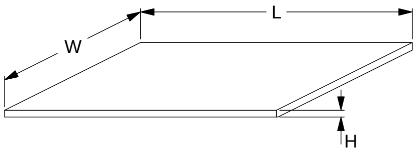
- An unfavorable routed signal may form a single winding of a coil and behaves like a transformer adding voltage somewhere.

- Depending on the circuit the impedance of a line has to fit to the rest of the system. (e.g. 50Ω)
- 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 influence of the disturbance depends on the used frequency, voltage, or current in the circuit. The goal is to minimize unwanted effects. That's by the way the reason why an 'Autorouter' is useless in most of the cases! You always have to keep full control! Next we will look at some of the unwanted effects....

9.3 PCB Properties

9.3.1 Resistance of traces



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

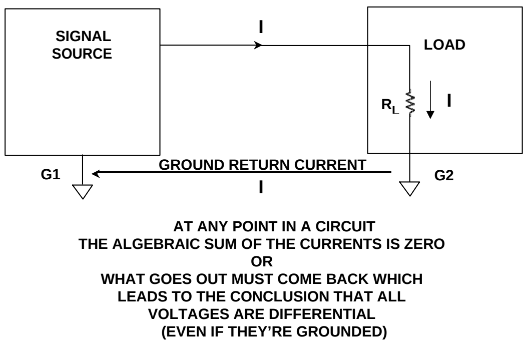
$$R = \frac{l\rho}{A}$$

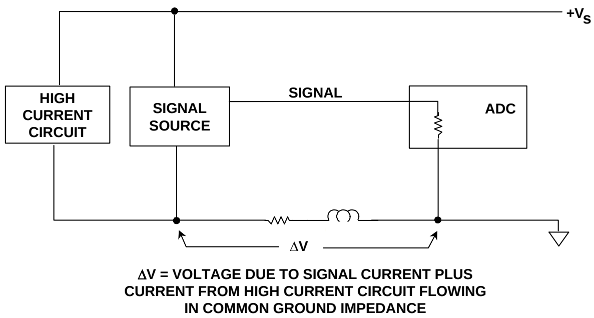
ρ = constant; for copper $= 1.78 * 10^{-6}\Omega$cm
l = length of trace
A $= H * W =$ cross section of trace

Example: $L = 50\,\text{mm}$, $W = 0.2\,\text{mm}$ then $R = 127\,\text{m}\Omega$ @ $25°$ C. Resistance changes by about $0.4\%/°C$. With higher currents you have to take care of the self heating of the trace!

Consider a 16-bit ADC with a $5\,\text{k}\Omega$ input resistance, driven by the above trace. The track resistance of $\approx 0.127\Omega$ forms a divider with the $5\,\text{k}\Omega$ load, creating an error. The resulting voltage drop is a gain error of $0.127/5\,\text{k} \approx 0.0025\%$, well over 1 LSB (0.0015% for 16 bits. And this ignores the issue of the return path!

Voltage Drop in Signal and Return Lines





Since the resistance in the signal path and between G1 and G2 is not zero, the voltage at the load is reduced!!

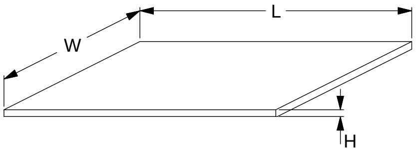
- Reduce R_{trace} in both paths
- Increase load resistance to reduce current
- Use feedback sense line

Other currents in ground path add additional voltage drop/ noise visible to the load.

- Use the already shown solution, additionally...
- Separate the ground path for high/ noisy return currents.

For both cases...
Since the voltage drops are common mode, use a differential design in the circuit. ('LinearDesign.pdf' page 12-13)

9.3.2 Inductance of traces

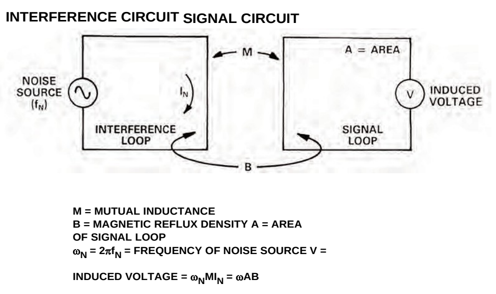


W= width
L = length
H = thickness typical $\approx 35\mu$m

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

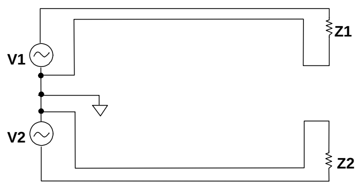
Example: $L = 10\,\text{mm}$, $W = 0.2\,\text{mm}$ then $L = 9.9\,\text{nH}$
The formula will given an approximate value, but is reasonably close to reality. At 10 MHz, an inductance of 0.9 nH has an impedance of 0.62Ω, and will induce $\approx 1\%$ error in a 50Ω system.

Mutual Inductance



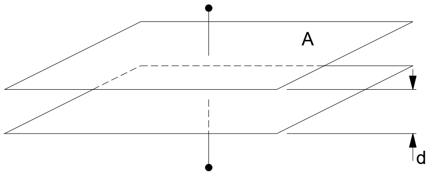
Mutual inductance is a magnetic coupling of coils, like in a transformer. In a circuit any KV loop forms a single winding of a coil. It receives and transmits magnetic fields. Voltage is induced in the receiving part.
The induced voltage is dependant on the current (strength of field) in the source,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

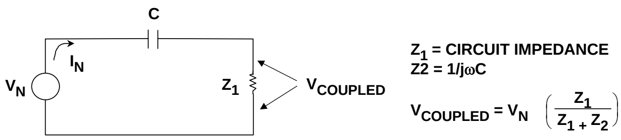
9.3.3 Capacitance of traces



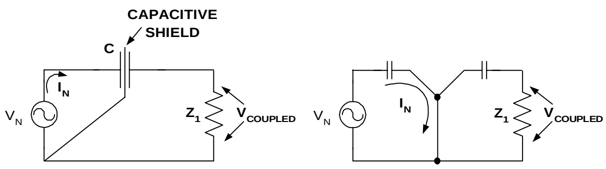
$A =$ plate area
$d =$ plate distance
$\varepsilon_0 =$ permittivity $8.8542 * 10^{-12}\,\mathrm{F/m}$
$\varepsilon_r =$ rel. permittivity ≈ 4.7 for FR4 material

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

A 100 mm long trace with 0.25 mm width has $\approx 0.7\,\mathrm{pF}$



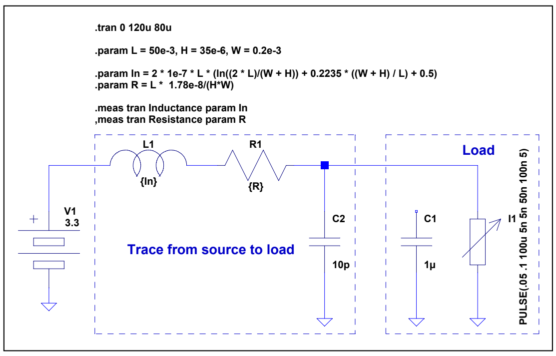
Replacement circuit for two coupled traces. C is the stray capacitance, V_{in} is a source, and Z_1 is the sink. The voltage divider formula gives the noise voltage.

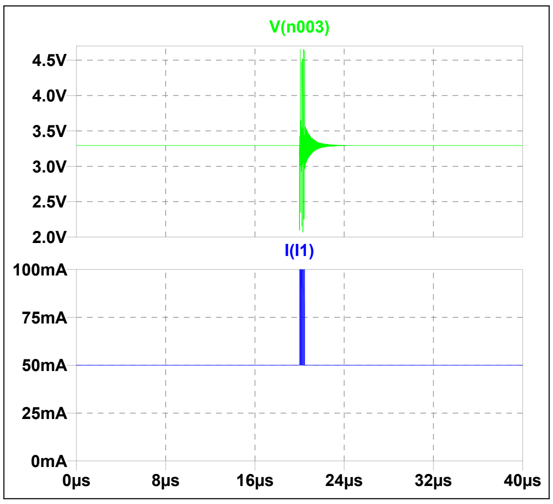


A ground plane ('Faraday shield') inside the PCB will eliminate the effect. The noise is shorted against ground in this case.

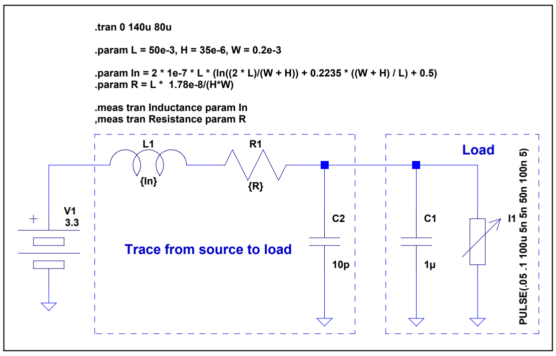
9.3.4 Example - Influence of Traces to current surges

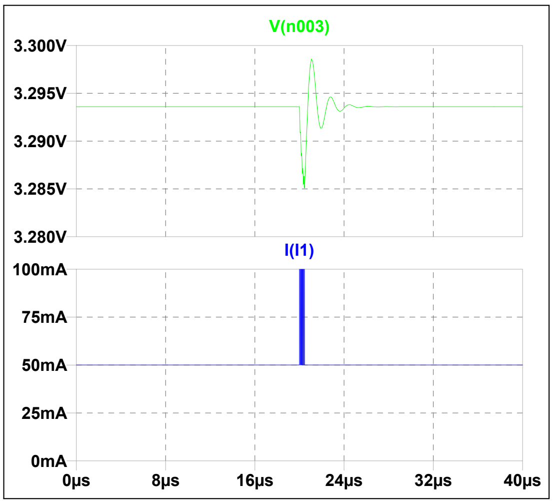
The simulation below shows how a poorly routed track and missing components can affect the voltage on a component. The length of the trace is assumed to be 50 mm and the width 0.2 mm. The thickness of the copper layer is 35μm as standard. The above formulas are applied. Only the inductance and the resistance are used. A freely laid conductive path without an opposite pole does not generate any capacitance. The $\approx 10p$F in the simulation is the capacitance of the input pin of the load.





The first example has no buffer capacitor close to the input pin. The pin has a continuous input current of $50m\,\mathrm{A}$. But then there are short load pulses like it happens in digital circuitry. It is clearly visible that the voltage at the load oscillates by $\approx 1\,\mathrm{V}$. Depending on the circuit that might mean malfunction!





The second example connects a capacitor to the load. Now the voltage varies only by some millivolt. The size of the capacitor depends on the needed current and the number of components. Of course also the physical size of the traces and the construction of the PCB can influence the behavior. It is only important that the buffer capacitors are placed very close to the supply pin of the load. This is to avoid influence of the trace.

9.3.5 Other Mechanical or Electrical Requirements

The placement of the components naturally depends on their size. Conductor paths and soldering areas must be producible and accessible for machining. The mechanical requirements sometimes conflict with the electrical requirements! Here are some points to consider for placement and routing.

- There are always mechanical requirements from the board manufacturer that must be met. For example, minimum distances between copper surfaces, minimum track widths, minimum drill hole sizes. Normally, each manufacturer issues a data sheet with these parameters.
- For large components, care must be taken to ensure proper fastening. The mechanical stability must be ensured. This means that they may have to be screwed or glued. Of course, space must be provided for this.
- When connecting a conductor track to a solder point, the size ratio of the two must be taken into account. The width of the trace must not be too narrow or too wide in relation to the size of the pad. An unfavorable size ratio can lead to problems in the manufacture of the PCB and also make soldering difficult.
- In addition to the aforementioned effects between the conductor tracks, dielectric strength must of course also be taken into account. The distance between

two live areas must be large enough. It is determined by the circuit board itself (e.g. protective lacquer), the layer within the circuit board and the ambient conditions such as air pressure!!!

- The solder pads on the board have to fit the element pads. In case of SMD parts it is clear that a small pad makes it impossible to solder. But a pad that is too large also makes soldering difficult or impossible. For round pads with drill hole it is similar. The soldering point will not be reliably producible. The size of the pads must be adapted to the soldering procedure.

This list is just an overview! There are some more things to take care about, but that will be to much for the given time.

9.4 Routing

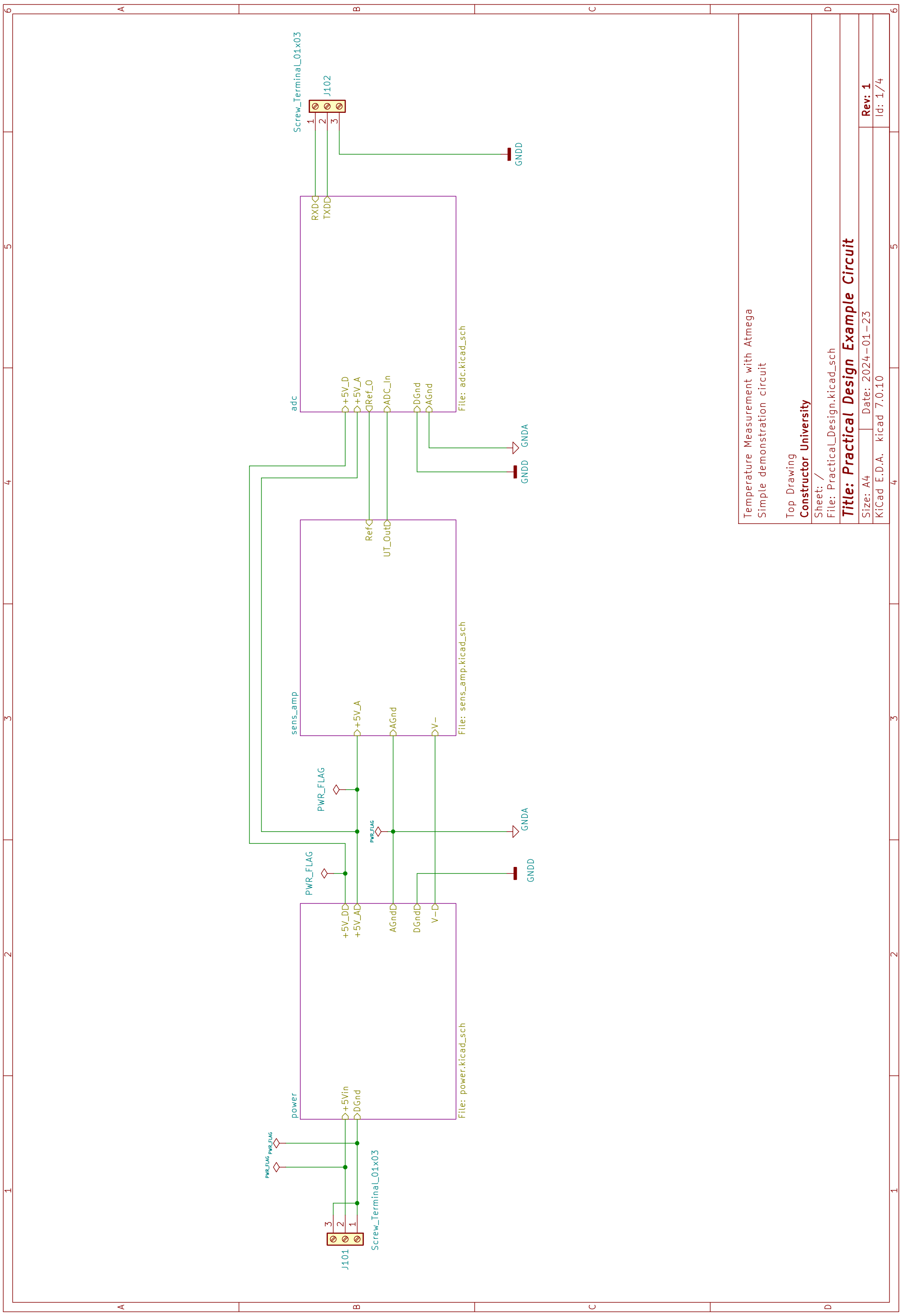
Now we can switch to the circuit board editor. Usage is similar to the schematic editor. Use the 'Getting started in KiCad' and the different KiCad help files for detailed description of the functions!

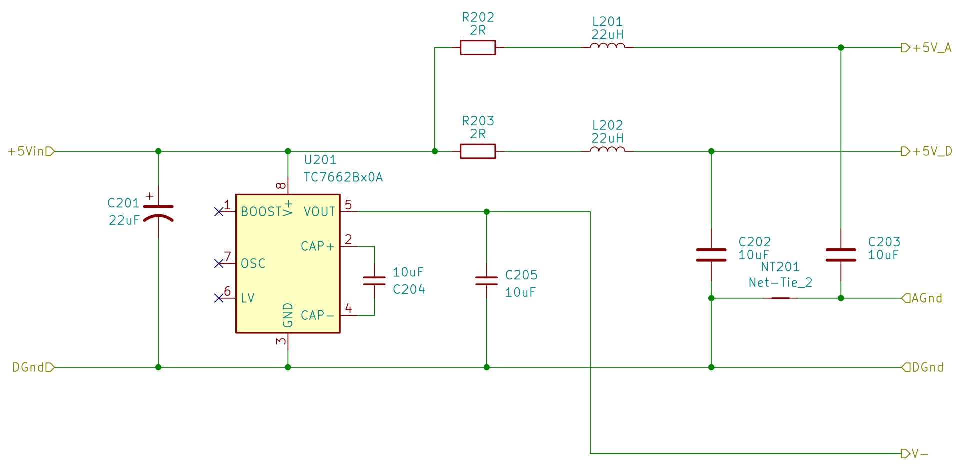
- Open the PCB editor. An empty sheet appears!
- Use the function 'Board Setup' from file menu. Remove all unused layers from the PCB. Under 'Design Rules/Pre-defined Sizes' define some track with. 8, 16, 32, 60, and 80 mil.
- Draw the specified mechanical shape/ dimensions of the circuit board on the 'EDGE.Cuts' layer.
- Import the net list from the schematic editor. All components will appear somewhere stacked connected via air wires.
- Are there any specifications where to place the I/O connectors/ contacts? Place the connectors and contacts. Fix the position!
  Use the example pictures from the manual as guideline.
- Use the defined functional blocks as a guide to place the components. Separate analog and digital sections.
- Final check of the mechanical layout. Are the device packages ok?

Now you have a board with (pre-placed!!) components. All pins should be connected via air lines and appear as short and 'unscrambled' as possible. The final step now is to route the board. The placement can still change... but remember the requirements!

10. Calculations, Examples

10.1 Circuit diagrams





Temperature Measurement with Atmega
Simple demonstration circuit

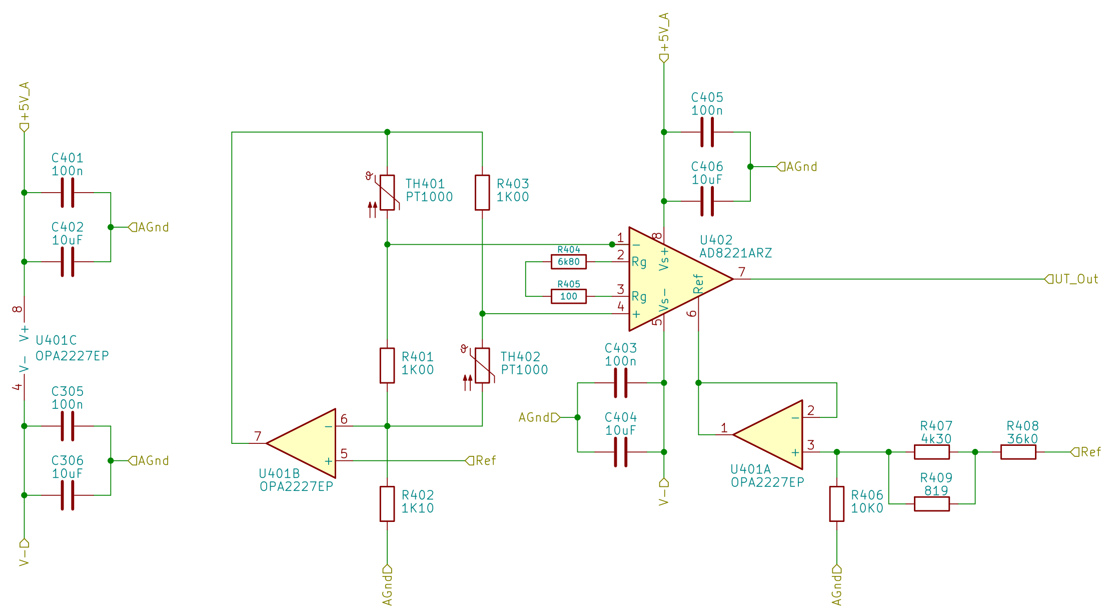
PowerSupply
Constructor University

Sheet: /power/
File: power.kicad_sch

Title: Practical Design Example Circuit

Size: A4 Date: 2024-01-23 **Rev: 1**
KiCad E.D.A. kicad 7.0.10 Id: 2/4

Gain = 8.1633
Gain resistor theoretical = 6896.30 used R_G = 6900
Offset resistor theoretical = 36688.66 used R_O = 36687.97



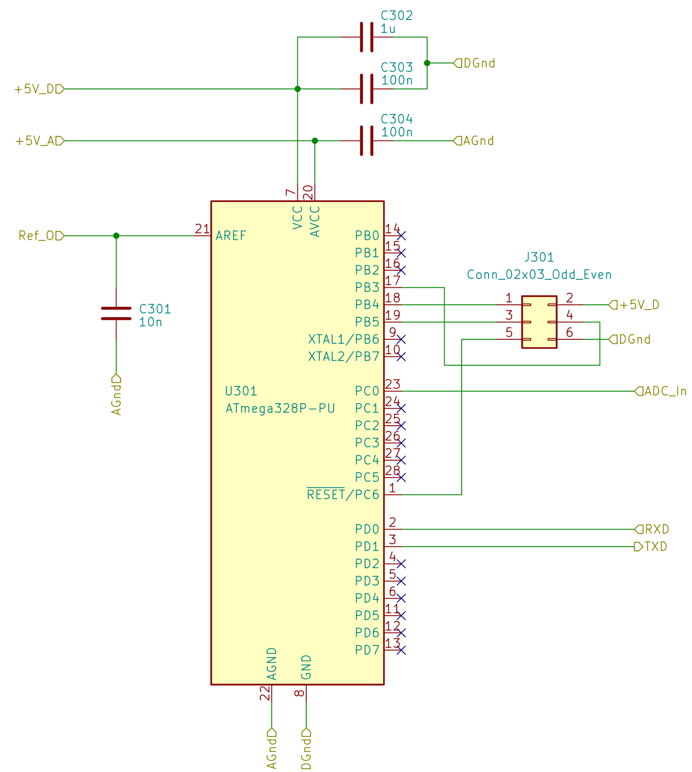
Temperature Measurement with Atmega
Simple demonstration circuit

Sensor/ Analog Circuit
Constructor University

Sheet: /sens_amp/
File: sens_amp.kicad_sch

Title: Practical Design Example Circuit

Size: A4 | Date: 2024-01-23 | **Rev: 2**
KiCad E.D.A. kicad 7.0.10 | Id: 3/4



Temperature Measurement with Atmega
Simple demonstration circuit

ADC/ Digital Interface
Constructor University

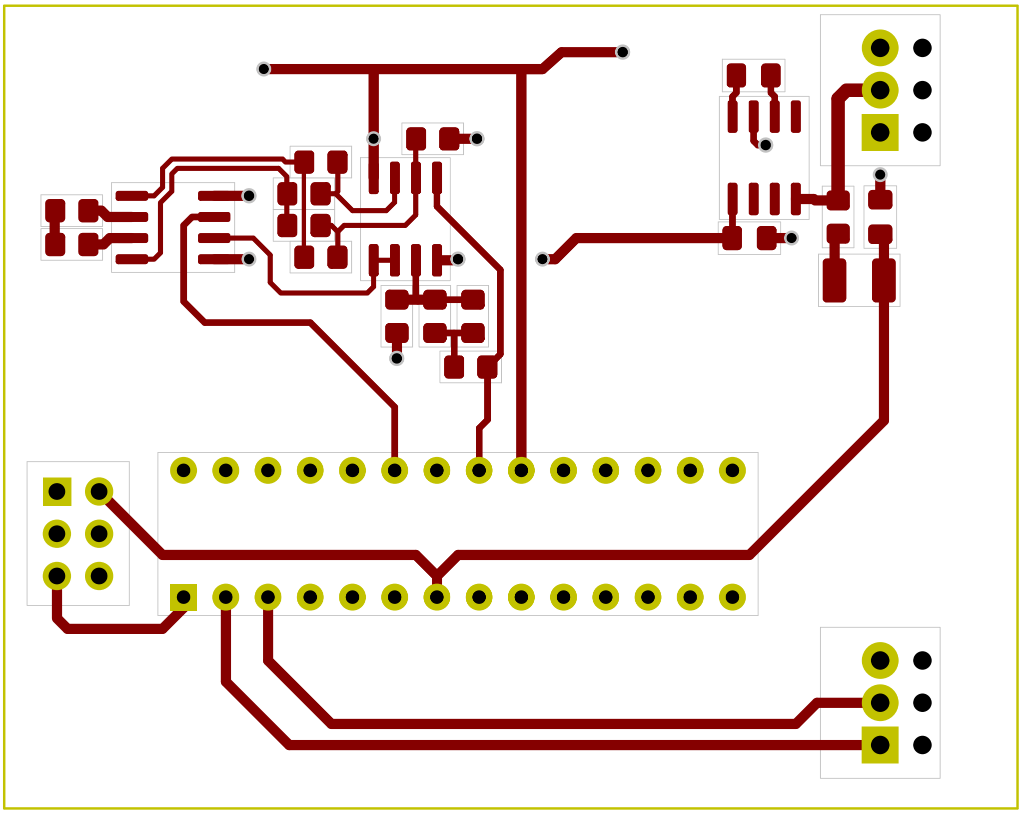
Sheet: /adc/
File: adc.kicad_sch

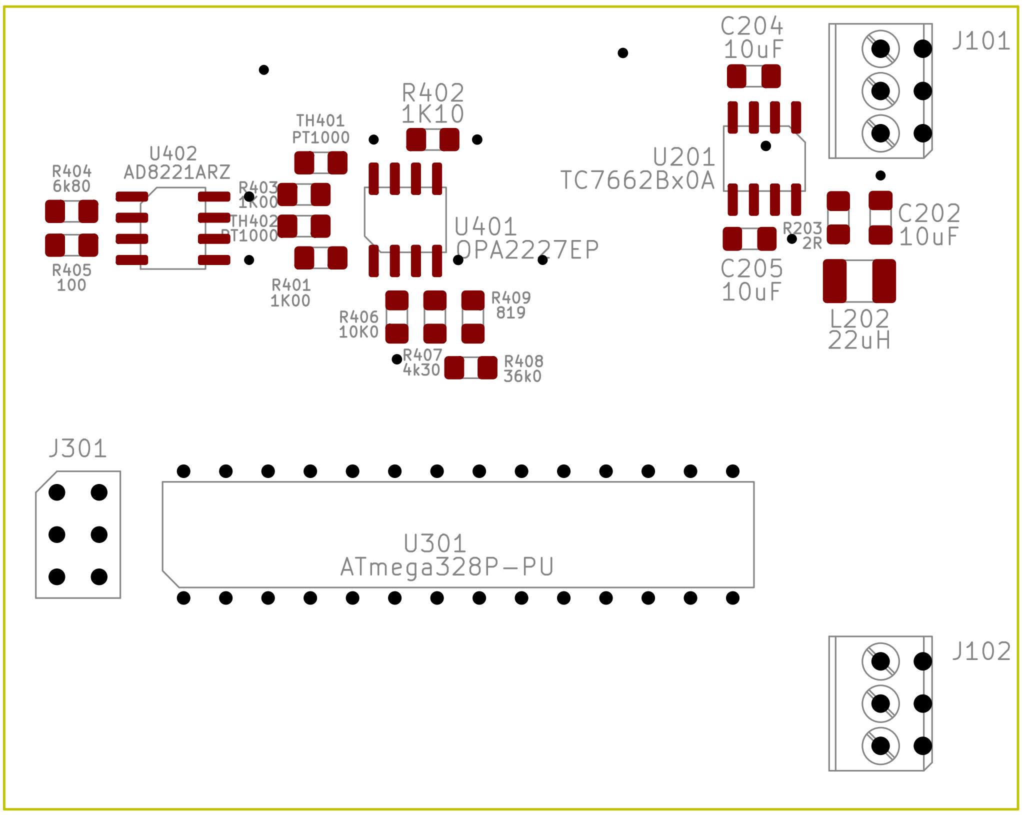
Title: Practical Design Example Circuit

Size: A4	Date: 2024-01-23	**Rev: 1**
KiCad E.D.A. kicad 7.0.10		Id: 4/4

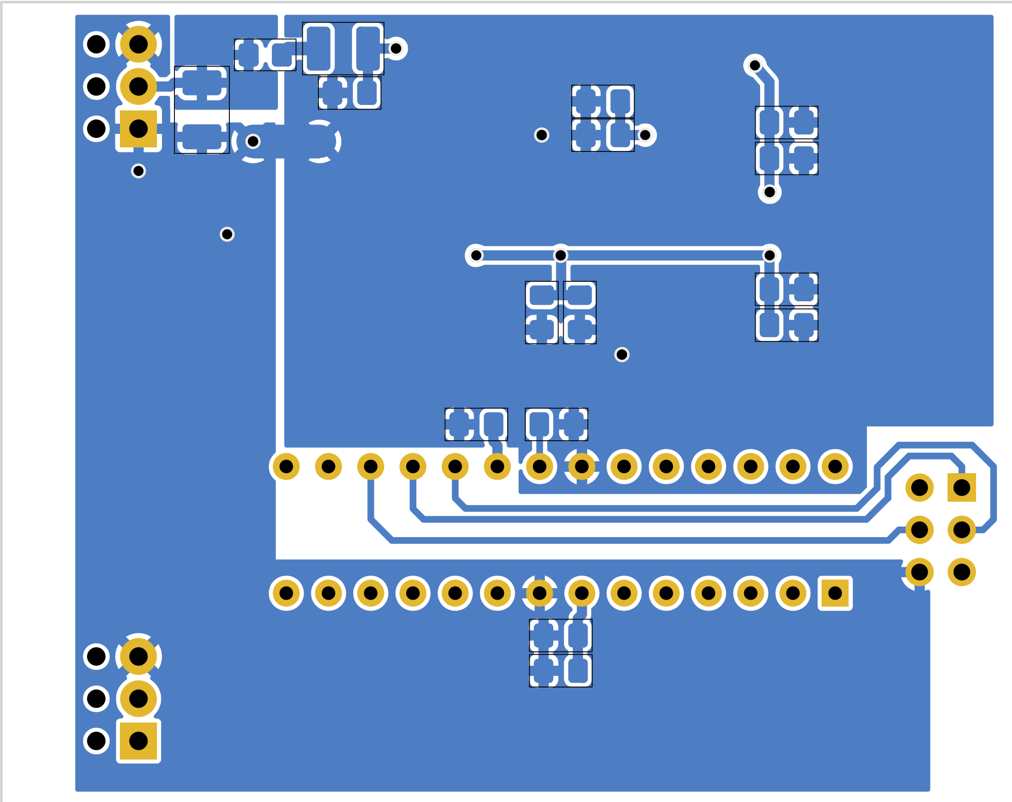
10.2 Example PCB

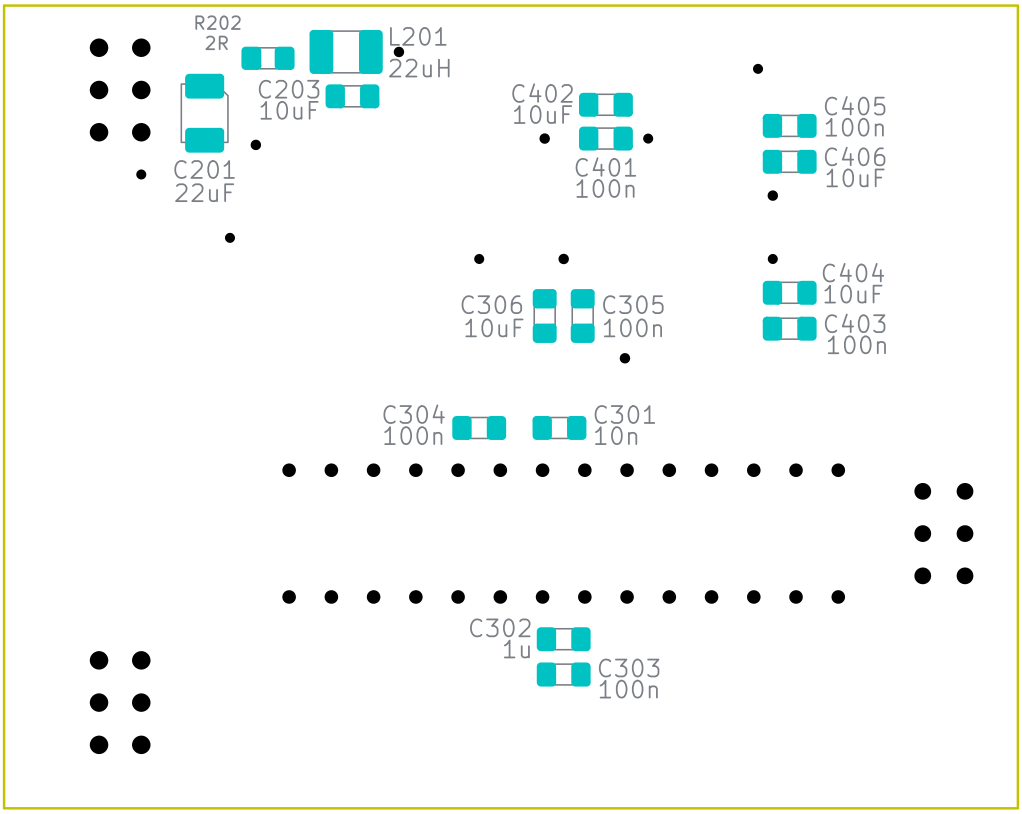
10.2.1 Top view - traces and placing



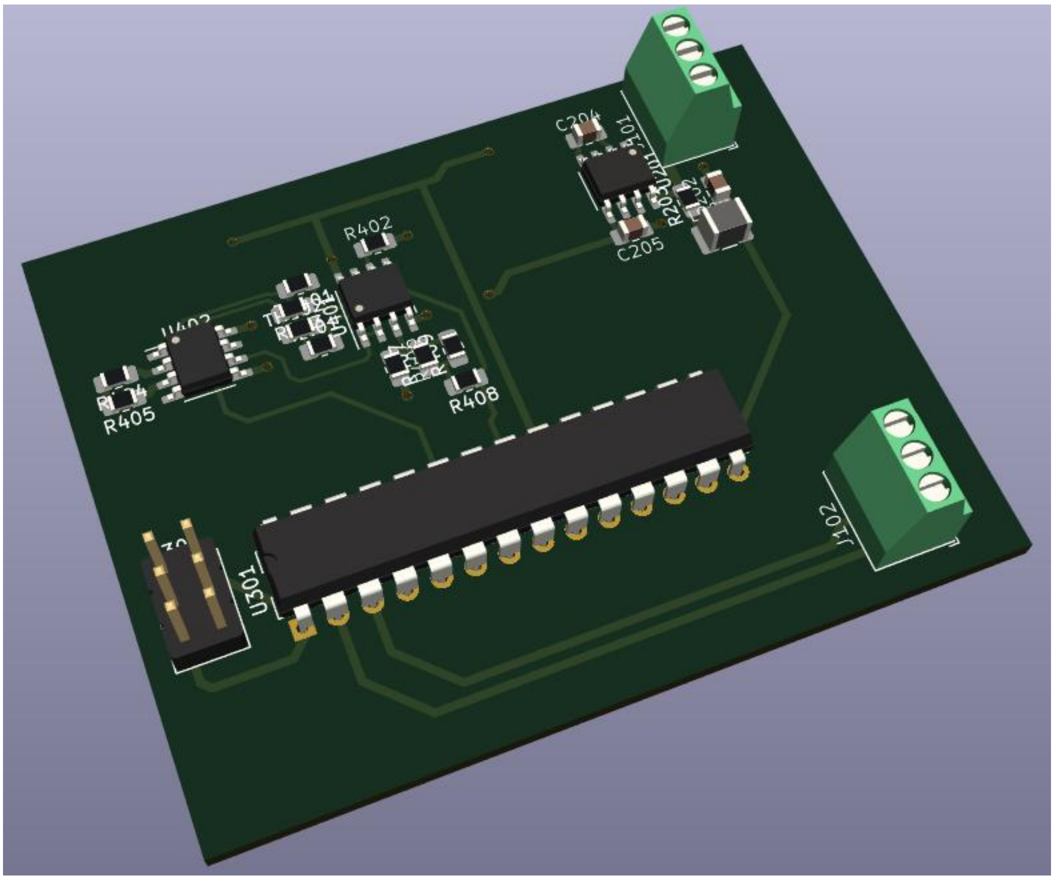


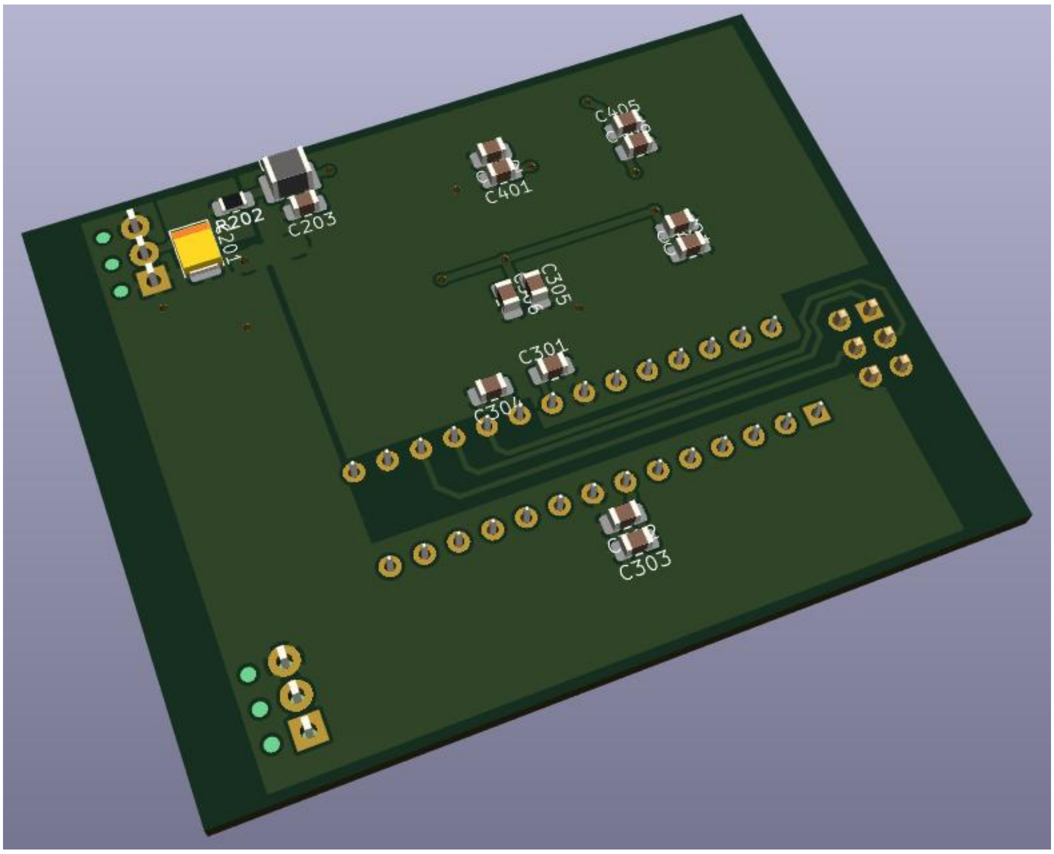
10.2.2 Bottom view (mirrored) - traces and placing





10.2.3 3d View of the Board





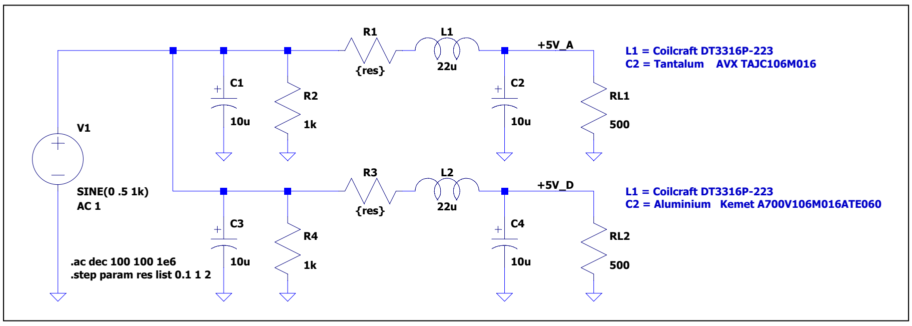
10.3 Explanations and Calculations

10.3.1 Check / Simulate the Power Circuit

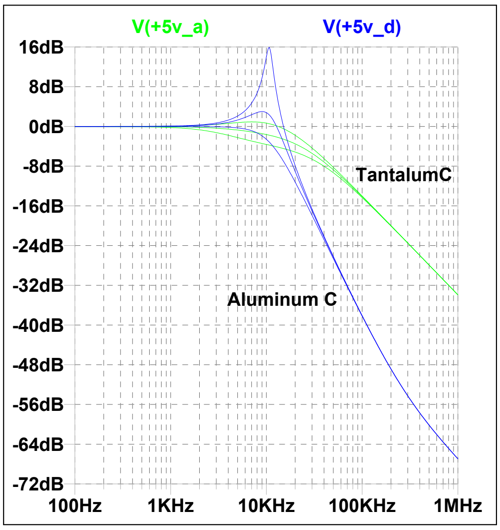
1. Two separate and filtered voltages are required. The 5V supply does not need to be further regulated. A so-called voltage inverter is used for the negative voltage. This converts the positive input voltage into a (slightly lower) negative output voltage. Generally speaking, the power supply does not have to provide exact values.

2. The positive supply voltages are filtered to suppress power supply disturbances.

For the negative part we can just take the circuit given in the data sheet and no simulation is needed.
The filter circuits must be checked. The coil and the capacitor influence the behavior. The selected coil should have a high quality and the capacitor a low ESR (see the component section!). This is a LTSpice simulation of the filter:



This is the result showing the magnitude at the output:



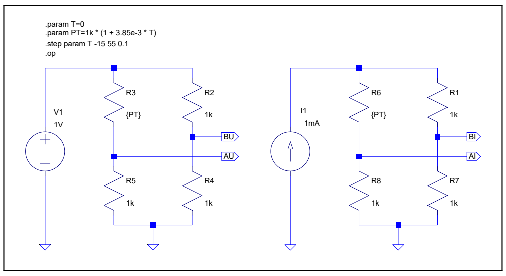
The case with the aluminum capacitor is better. The reason for this is the much lower ESR!! So for C202 and C203 10μF, 16V KEMET capacitors are chosen. The case size for these capacitors is 2917 (in mil) or 7343 (in mm).
The resistor in series with the inductor dampens the overshoot close to the cutoff frequency. 1Ω is approximately the critical damped case. A damped case is better, therefore a 2Ω resistor is chosen.

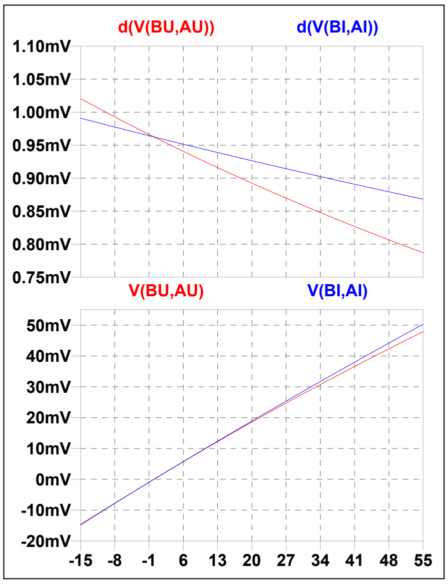
10.3.2 Check / Simulate the Sense_Amp Circuit

Thoughts on the function of the Wheatstone bridge

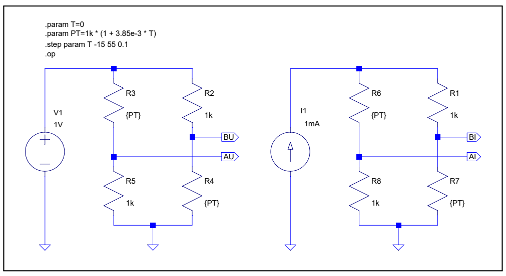
A Wheatstone bridge with one or more PT1000 resistors is to be used as the measuring circuit. To keep the effort for the amplifier and the AD conversion simple the output voltage should be as high and linear as possible. Four cases will be simulated. The bridge is used with one or two sensors and the supply is a voltage or a current source. One the web-site there is additional literature about the behavior of different Wheatstone bridge configurations!!

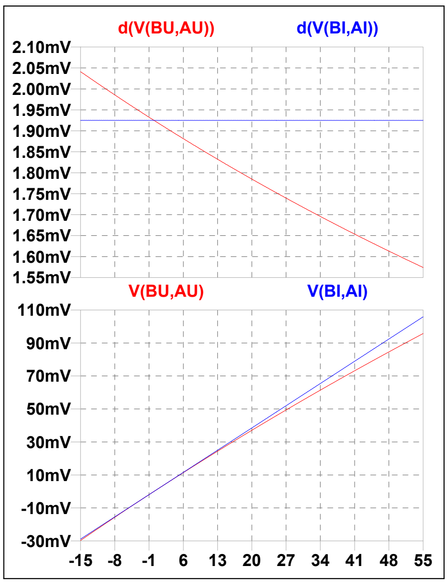
1. Wheatstone Bride with one PT1000 sensor supplied by a voltage and a current source. Temperature range is from $-15 > +55°C$.:





2. Wheatstone Bride with two PT1000 sensor again supplied by a voltage and a current source. Temperature range is from $-15 > +55°C$.:

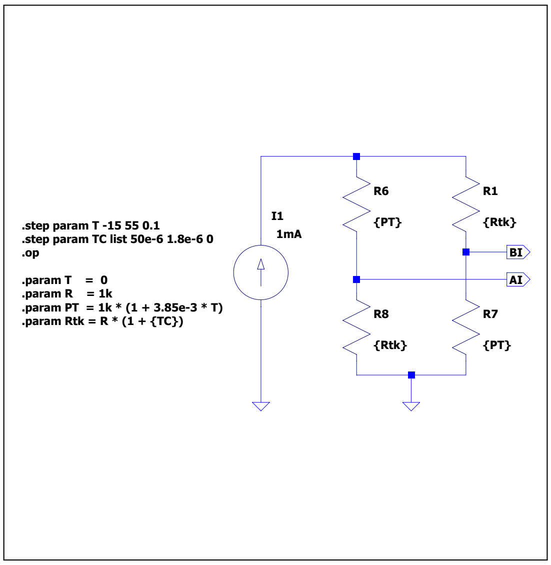


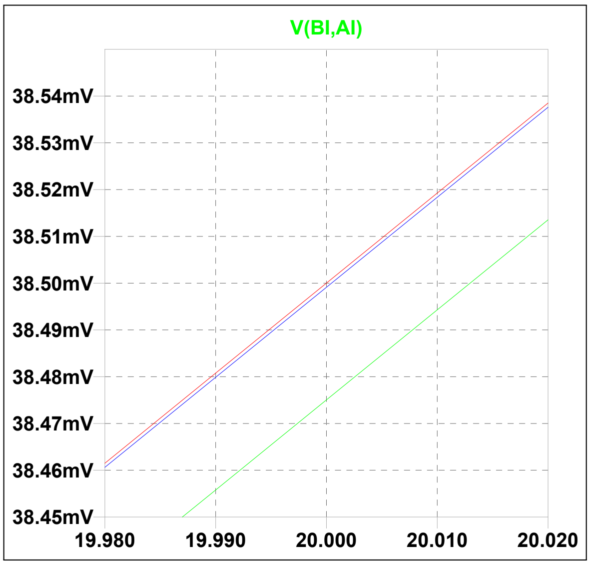


To determine the linearity of the output voltage over the temperature range the derivative of the output is shown (LTSpice d() function). The output differential voltage is clearly non linear when using a single sensor like in the first case. In the second case the bridge circuit supplied by the current source is linear. Also the output voltage doubles because of the two sensors.

Selection of resistors for the bridge

The fixed resistances of the bridge should be accurate and exactly the same. The values should remain stable over the desired temperature range. I.e. the TC coefficient should be low. The following simulation shows the behavior at 0ppm/°C, 50ppm/°C, and 1.8ppm/°C.





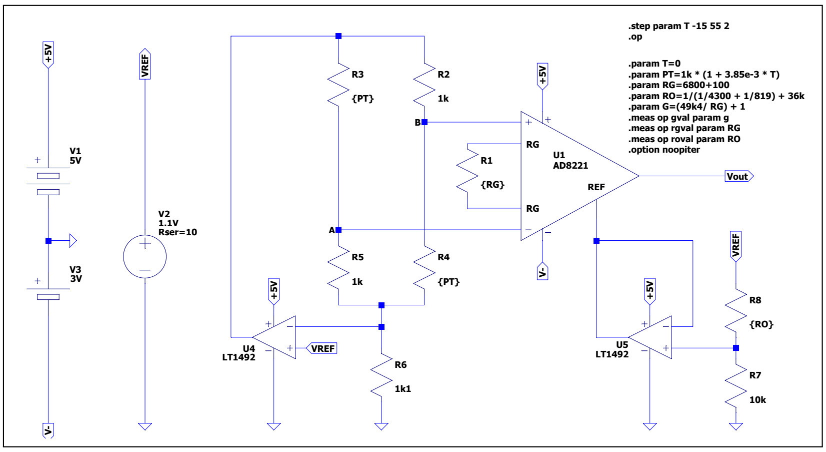
0ppm/°C would be optimal but is impossible! 50ppm/°C is usually a good TC. 1.8ppm/°C is the chosen TC from the selection above. In the simulation, the red line is the ideal case. At 1.8ppm/°C (blue line) the difference is very small. At 50ppm/°C (green line) the difference of the output voltage is already large. The error in this case is $\approx 0.2\%$.

Bridge circuit, Current source, and Amplifier

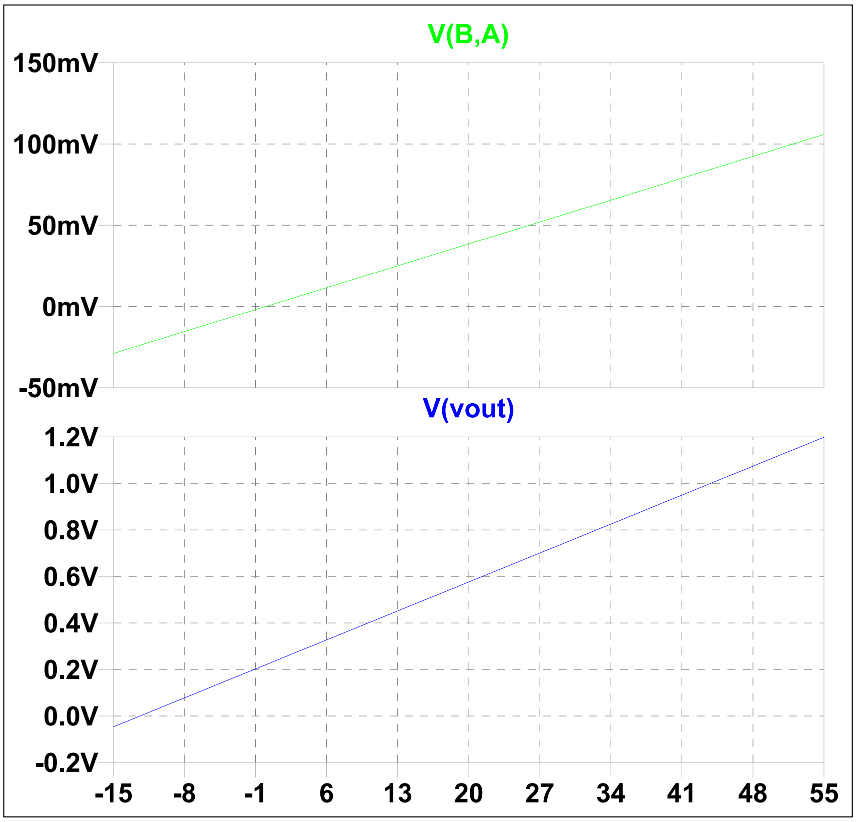
- As a result of the simulation a Wheatstone bridge with two sensor elements supplied by a current source is used.
- Op-amp U401B, R402, and the bridge circuit are a non inverting amplifier. The bridge is in the feedback path. If a reference voltage is connected to the non inverting input of the Op-amp the bridge is supplied with a constant current of $V_{ref}/R402$. In our case 1mA.
- The bridge output is amplified by an instrumentation amplifier. It should convert the resulting output voltage of the temperature range $-15 > +55$°C to $0 > 1.1$V.
  1. The voltages at the bridge are all positive. But because the output works differential, the input receives a two-pole voltage! Due to the uni-polar input of the ADC we have to avoid negative voltages. For this purpose there is an input at the amplifier with which an offset can be added to the output. This offset is set with R406, R407, R408, and R409 and the reference voltage.
  2. R404+405 set the gain of the amplifier and determine the amplification.
  3. Since the reference voltage cannot supply enough current for the bridge and the offset, a voltage follower U401A is used to amplify it.

Calculations and Simulation

First a complete simulation of the circuit with 'random' values for Gain (6000Ω) and offset (40 kΩ).



The simulation shows the output voltage of the Wheatstone Bridge and the output of the differential amplifier over the requested temperature range $-15°$ to $+55°C$.



The output of the simulation is not exactly what we need! It seems to be linear but the graph doesn't start at 0V and the max. value is not 1.1V.
The differential input voltage has to be amplified and an offset has to be added.

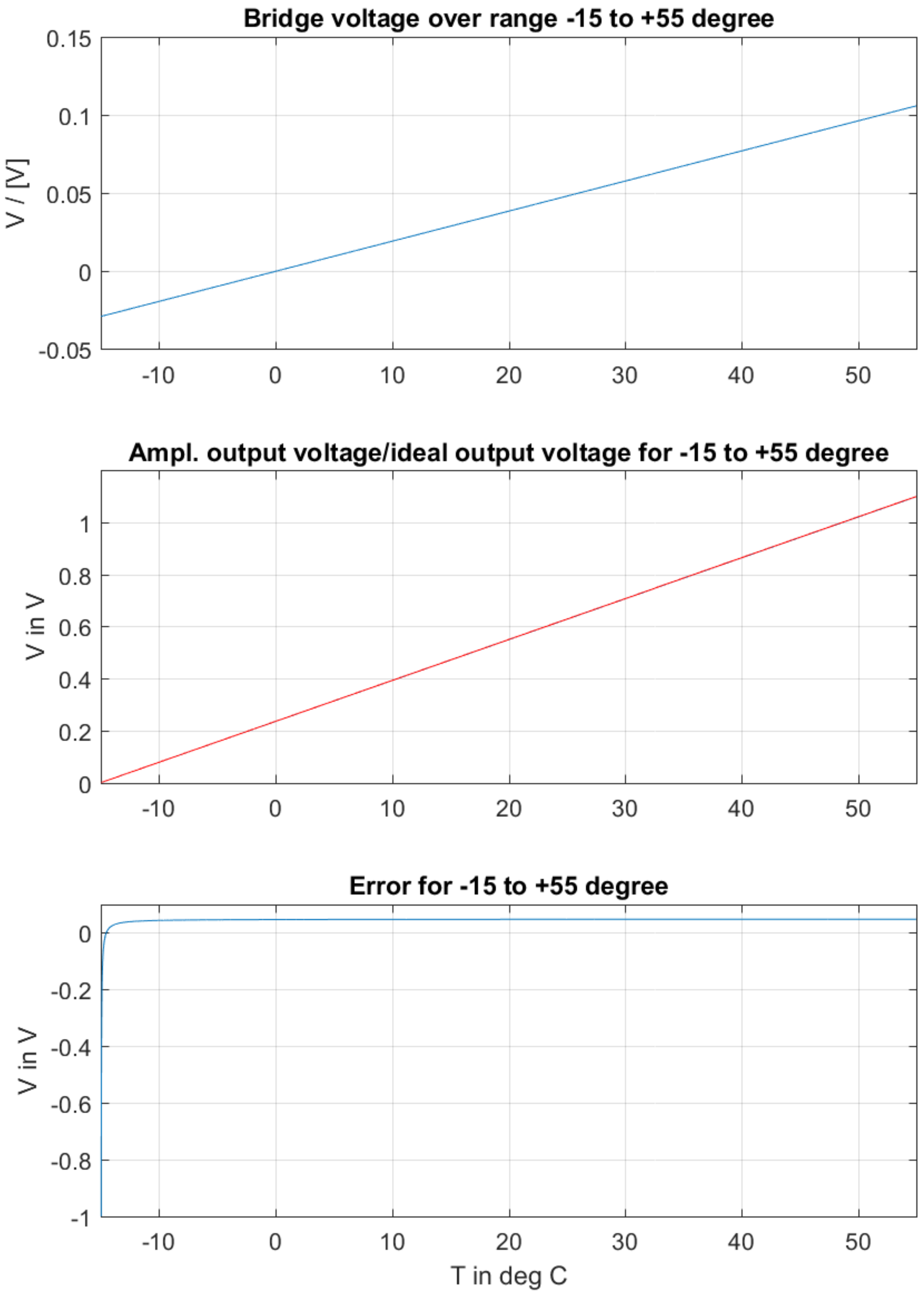
1. The overall voltage is $V_{\pm} = V_{max} - V_{min}$. $V_{\pm}$ determines the gain for the instrumentation amplifier.

2. V_{min} determines the offset value.

Output from Matlab:

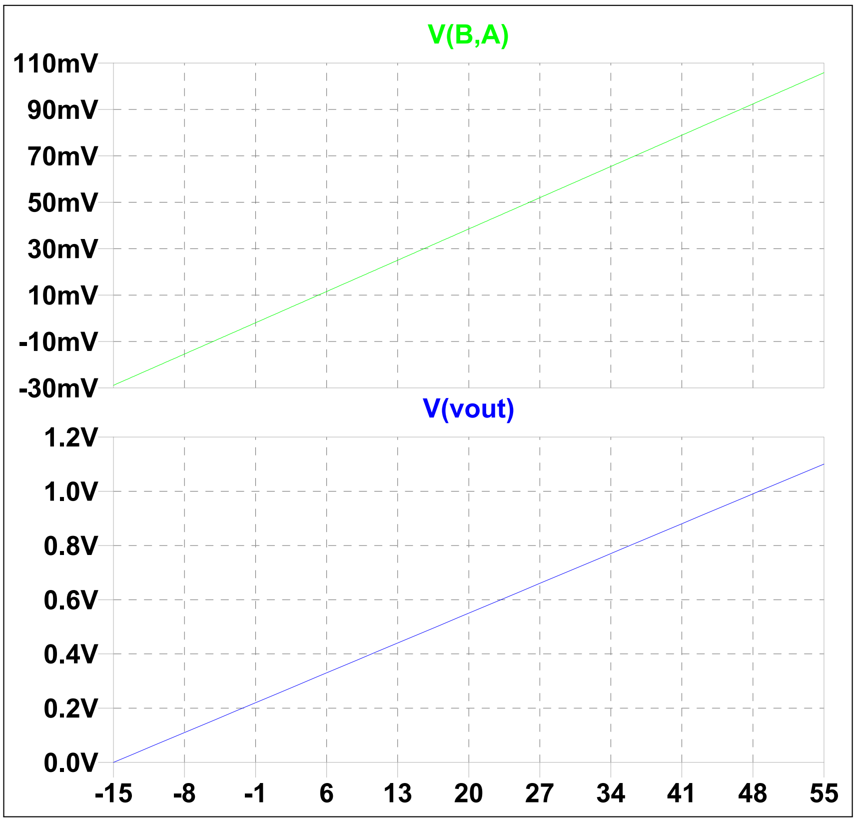
```
Bridge Vmin@-15Deg = -0.028875V -- Vmax55Deg =  0.105875V
                                -- Vdiff =  0.13475V

Calculated Gain is G = 8.1633 -- calculated resistor R_G =  6896.30

Chosen resistors from E24 series R404 = 6800Ohm, R405 =  100Ohm
Gain resistor is 6900Ohm results to G = 8.1594

Calculated offset for the amplifier is 0.2356V
Calculated offset voltage divider - R_lo = 10000.00 - R_hi = 36688.66

Chosen resistors from E24 series R407 = 4300.00Ohm,
                                 R408 = 36000.00Ohm, R409 = 819.00Ohm
Selected R_hi = 36687.97 results to offset voltage 0.2356V
```

The plotted results:



The calculated resistors are nearly available!! So practical output will become more or less accurate. Only the tolerance of the components will change the results.

Now the complete LTSpice simulation of the circuit with calculated values. The circuit is the same from before. Only the resistors are changed.



LTSpice gets of course the same result as MatLab. The circuit is operational now and all missing values are known.

The used MatLab code:

```
%==================================================================
% Calculation of the Amplifier range and the offset. -- .01.2024
%==================================================================
% first clean up and format output
clc; clear; close all; format short eng

% -------------------------------------------------------
% Constants:
u_ref    = 1.1;         % 1. Reference voltage
r_ref    = 1100;        % 2. Resistor determing I-REF
i_ref    = u_ref/r_ref; %
t_ref    = 0;           % reference T of PTC
r_0      = 1000;        % 3. 0 Deg value of -all- resistors
alpha    = 3.85e-3;     % 4. Temperature coefficient of PT resistor
t_min    = -15;         % 5. min temperature to be converted = 0x000
t_max    = 55;          % 6. max temperature to be converted = 0x3ff
r_div_l  = 10e3;        % 7. Lower resistor in offset voltage divider

% -------------------------------------------------------
% length of all vectors; including min/ max values
pts = ((t_max - t_min) + 1) * 100; %

% -------------------------------------------------------
% resistance of the PT1000 over range
temp = linspace (t_min, t_max, pts);
r_pt = r_0 * (1 + alpha * (temp-t_ref));

% -------------------------------------------------------
% bridge voltages over range and % max. voltage difference
u_bridge = i_ref/2 * (r_pt - r_0);
u_delta = u_bridge(pts) - u_bridge(1);

% G is amplification to fit the bridge value into the ADC range
% determine the gain resistor
G_th = u_ref / u_delta;
r_gain_th = 49.4e3 / (G_th-1);      % for AD8221

% select resistors from standerd series calculate resulting gain
R404 = 6800;  R405 = 100;  r_gain = R404 + R405;
G = 1 + (49.4e3 / r_gain);

% -------------------------------------------------------
% Determine the offset for the amplifier. Offset should be
% a little bit higher than the max. negative output
u_off_calc = (0 - u_bridge(1)) * G;
```

```
% determine offset voltge divider. Lower resistor is set to
% 10K. Upper one has to be calculated
i_off_div    = u_off_calc/ r_div_l;
r_div_h_calc = (u_ref - u_off_calc) / i_off_div;

%-------------------------------------------------------------
% Check calculation using standard resistors
% select resistors from standerd series (E24 3k20) ...
% voltage should become slightly higher then calculated
R407 = 4300;  R408 = 36e3; R409 = 819;
r_div_h = (1/R407 + 1/R409)^-1 + R408;
u_off   = u_ref * r_div_l / (r_div_h + r_div_l);

%------------------------------------------------------------
% ... that should be the output from the amplifier
% -15mV to average the error! Means reduce the offset voltage.
u_amp = (u_bridge * G) + u_off;

% calculate the ideal linear output
u_ad_lin = linspace (0, 1.1, pts);

%------------------------------------------------------------
% print the results
fprintf ('Bridge Vmin@%2gDeg = %9.6gV ', t_min, u_bridge(1));
fprintf ('-- Vmax%2gDeg = %9.6gV -- Vdiff = %8.6gV\n\n', ...
t_max, u_bridge(pts), u_delta);
fprintf (['Calculated needed Gain is G = %6.4f ' ...
'-- calculated resistor R_G = %8.2f\n\n'], G_th, r_gain_th);

fprintf (['Chosen resistors from E24 series ' ...
'R404 = %4gOhm, R405 = %4gOhm\n'], R404, R405);
fprintf (['Gain resistor is ' ...
'%4gOhm results to G = %6.4f\n\n'], r_gain, G);

fprintf ('Calculated offset for the amplifier is %6.4gV\n', u_off_calc);
fprintf ('Calculated offset voltage divider - R_lo = %6.2f - R_hi = %6.2f \n\n',
r_div_l, r_div_h_calc);
fprintf (['Chosen resistors from E24 series ' ...
'R407 = %6.2fOhm, R408 = %6.2fOhm, R409 = %6.2fOhm \n'], ...
R407, R408, R409);
fprintf ('Selected R_hi = %6.2f results to offset voltage %6.4gV \n', ...
r_div_h, u_off);

%------------------------------------------------------------
% plot the results
%
```

```
% plot the voltage over the bridge from t_min to t_max
subplot(3, 1, 1)
plot (temp, u_bridge); grid;
tit = sprintf ('Bridge voltage over range %+d to %+d degree', ...
t_min, t_max);
title (tit)
xlim ([t_min t_max]); ylim([-50e-3 150e-3]); ylabel('V / [V]')

% second figure with amplifier output and ideal output
subplot(3, 1, 2)
plot (temp, u_amp);
hold on;
plot (temp, u_ad_lin, 'Color',[1 0 0]); grid; % rot data2
tit = sprintf (['Ampl. output voltage/ideal output voltage for ' ...
'%+d to %+d degree'], t_min, t_max);
title (tit);
xlim ([t_min t_max]); ylim([0 1.2]); ylabel('V in V')

% third figure with error
err = (u_ad_lin(2:end) - u_amp(2:end)) ./u_ad_lin(2:end) *100;
subplot(3, 1, 3)
plot (temp(2:end), err); grid;
tit = sprintf ('Error for %+d to %+d degree', t_min, t_max);
title (tit)
xlim ([t_min t_max]); ylim([-1 0.1])
xlabel ('T in deg C'); ylabel('V in V')
```

Part III

Additional Information

A. Appendix

A.1 Books and other Tools

A.1.1 Book

- Sarma
- Floyd

A.1.2 Programs

LTSpice

LTSpice is a powerful, unlimited circuit simulator for analog circuits. It includes a graphical schematic capture interface and a waveform viewer to shows the results. It is freely available for Windows, Mac, and with an emulator also for Linux. Download link is *'from Analog Devices'*.

Octave

GNU Octave is a high-level language primarily intended for numerical computations. It is typically used for such problems as solving linear and nonlinear equations, numerical linear algebra, statistical analysis, and for performing other numerical experiments. It may also be used as a batch-oriented language for automated data processing. It is freely available for Windows, Mac, and Linux. Octave has been built with MATLAB compatibility in mind, and shares many features with MATLAB. Octave is available from *'https://octave.org/'*.

KiCad

KiCad is an open-source software suite for creating electronic circuit schematics, printed circuit boards (PCBs), and associated part descriptions. KiCad supports an integrated design workflow in which a schematic and corresponding PCB are designed together, as well as standalone workflows for special uses. KiCad also includes several utilities to help with circuit and PCB design, including a PCB calculator for determining electrical properties of circuit structures, a Gerber viewer for inspecting manufacturing files, a 3D viewer for visualizing the finished PCB, and an integrated SPICE simulator for inspecting circuit behavior. It is freely available for Windows, Mac, and Linux from *'https://www.kicad.org'*.

MatLab

MATLAB (an abbreviation of "MATrix LABoratory") is a proprietary multi-paradigm programming language and numeric computing environment developed by Math-Works. MATLAB allows matrix manipulations, plotting of functions and data, implementation of algorithms, creation of user interfaces, and interfacing with programs written in other languages. It is available for Windows, Mac, and Linux. As registered student you may download it and use the university based license.