PCB Design with KiCad

Introduction and Guidance

for Physicists at HIM

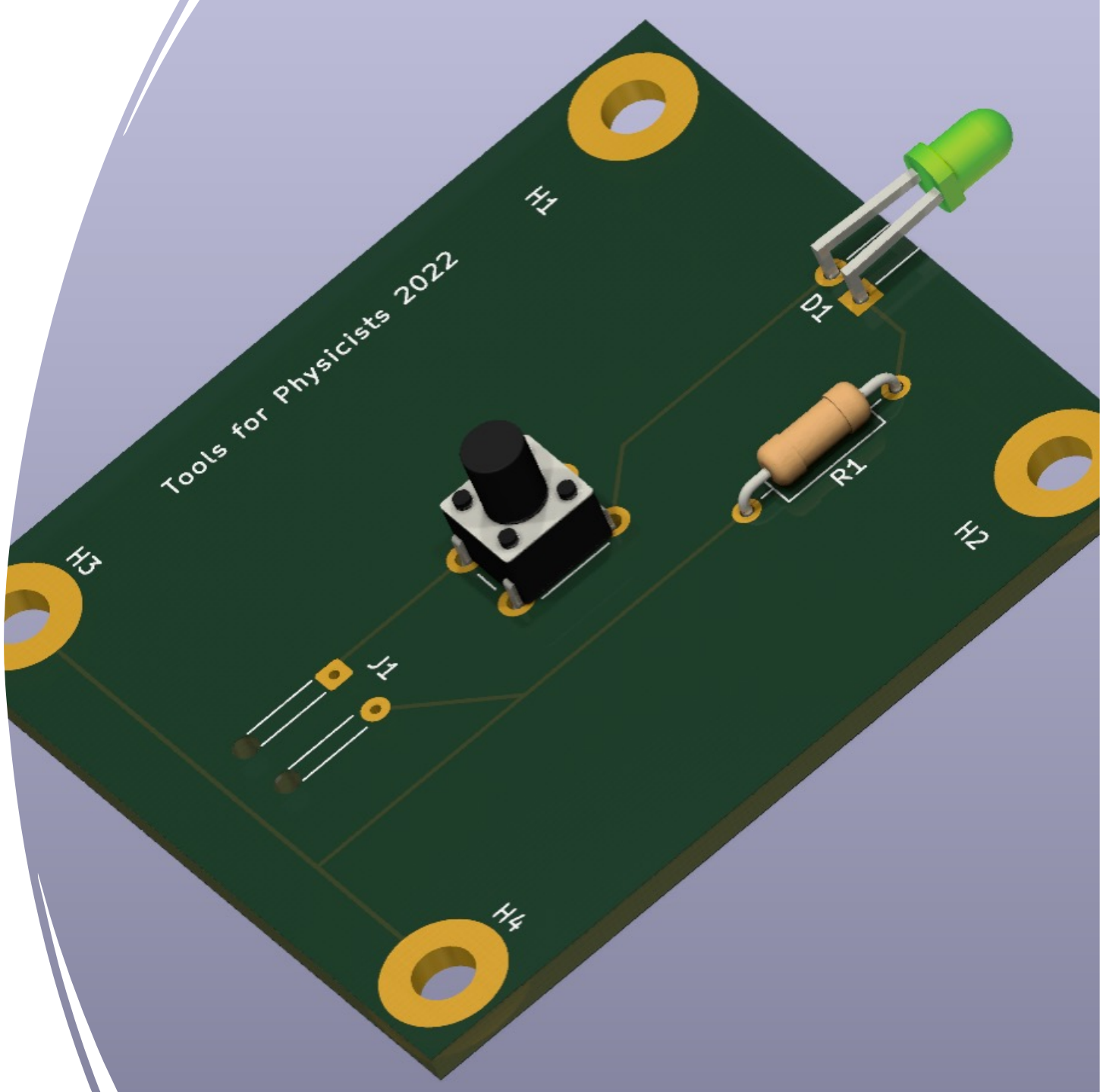
5.7.2023 – Peter-Bernd Otte - HI-Mainz

Our overall goal today

- Design and produce a printed circuit board (PCB)!

PCB:

1. affix electronic components
2. electrical connections



What is achievable? Do not start if…

If you strive for …

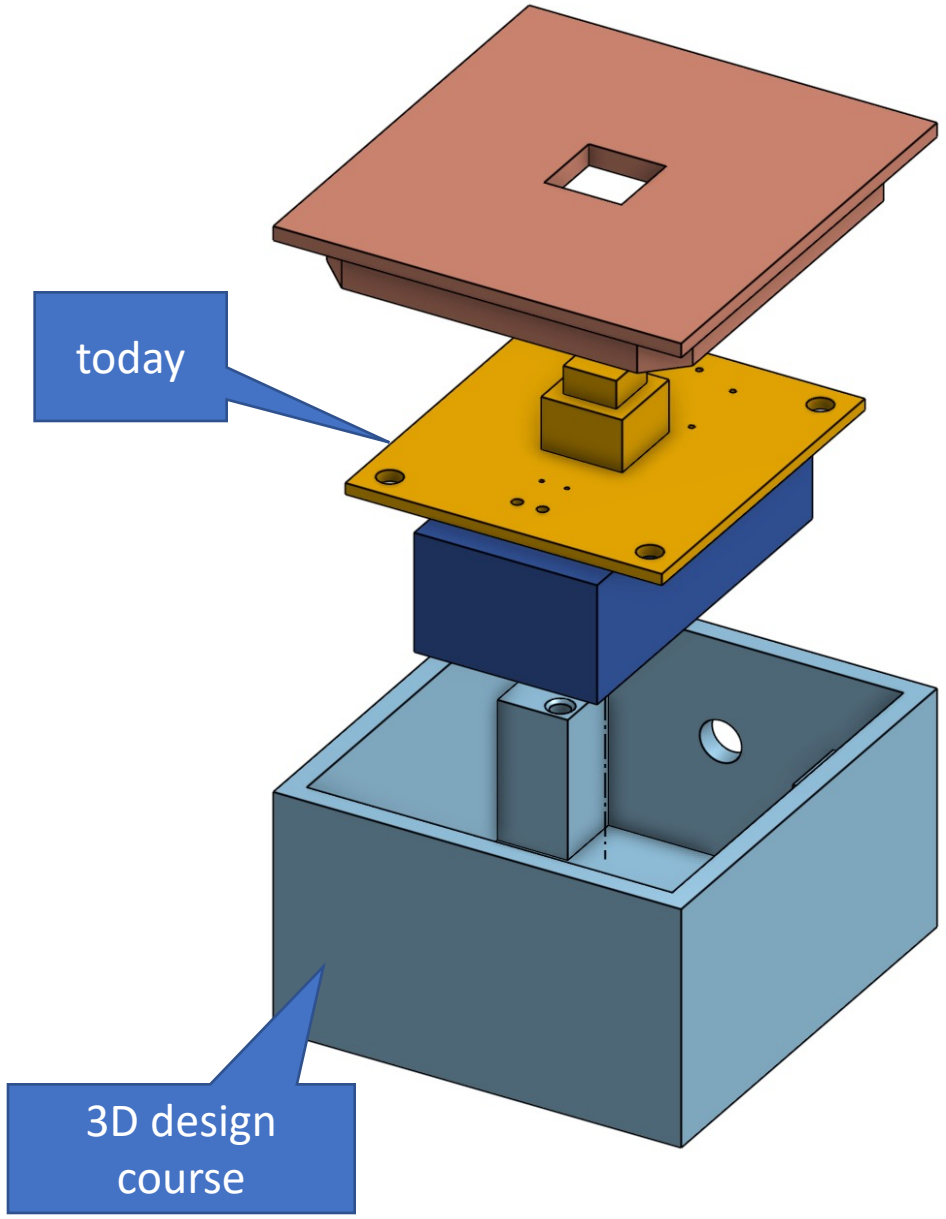
- high currents >1A → hot traces or contacts
- high voltages >42V → induce ventricular fibrillation
- high frequencies >20 MHz → impedance becomes important
- multi layer PCB >4 layers → complicated
- small / complicated parts → hard to solder
- large number of boards → usage of panels (German: “Nutzen”)

→ contact the electronics workshop at KPH (Igor Beltschikow and crew)

Focus today: slow control

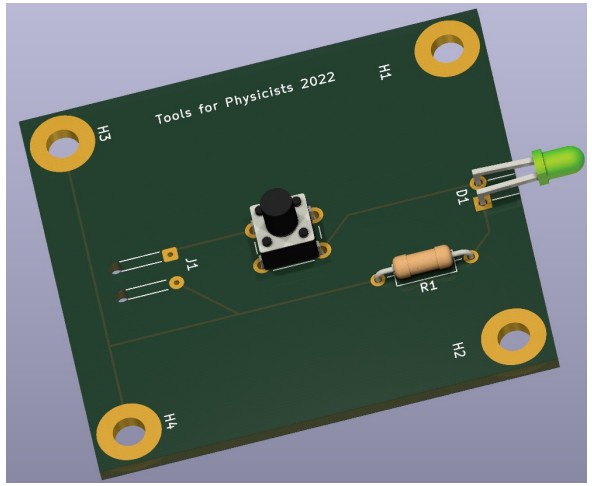
Today’s overview

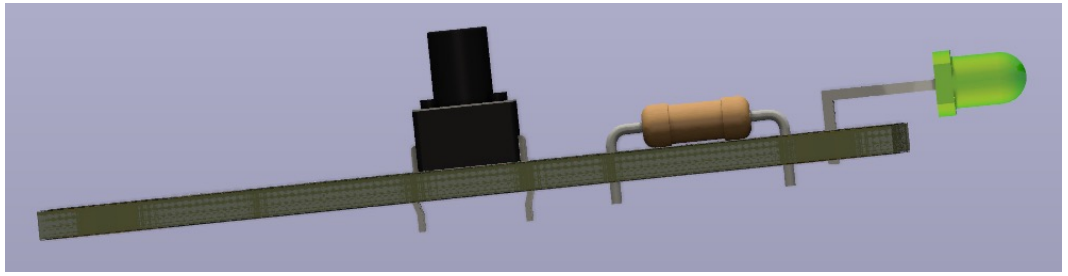
1. THT, SMD / Reflow soldering / PCB properties
2. PCB design software overview
3. KiCAD’s advanced functionality showcase
4. 1st KiCAD project: a flash light
5. custom symbols and footprints
6. bonus project: lab environment slow control

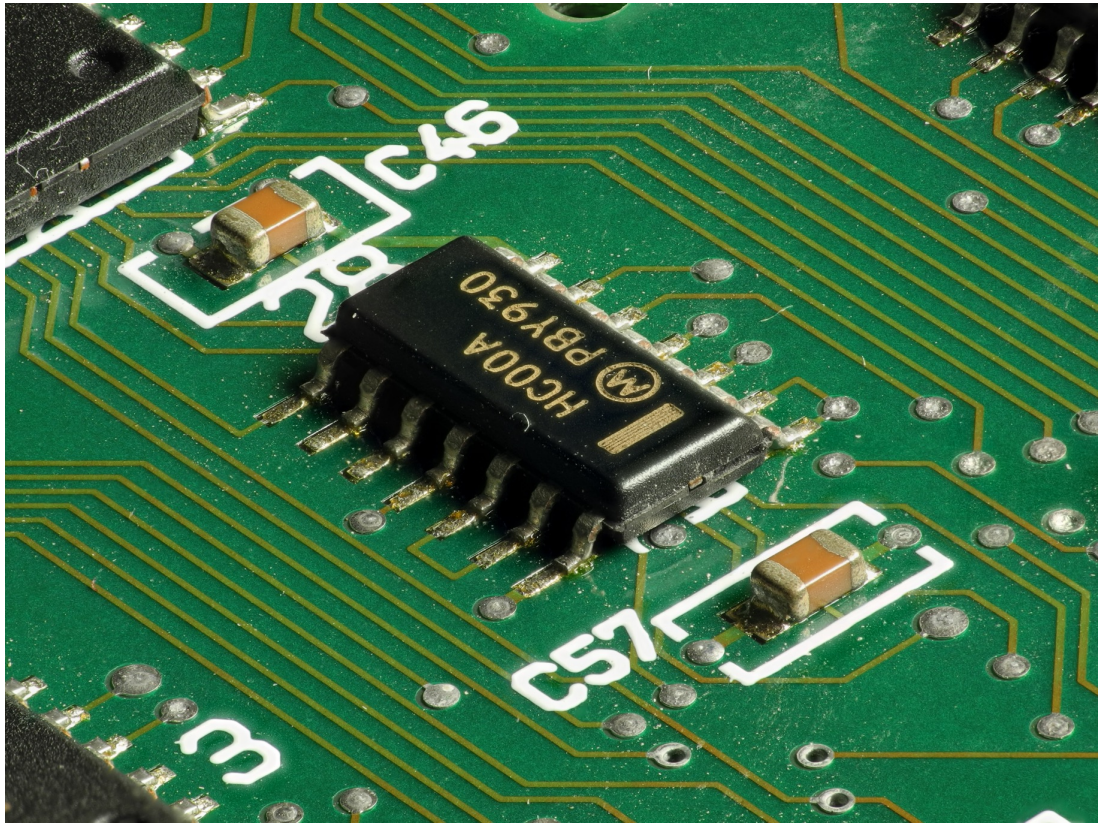


1) THT and SMD

- Through hole technology (THT) and Surface Mount Device (SMD)

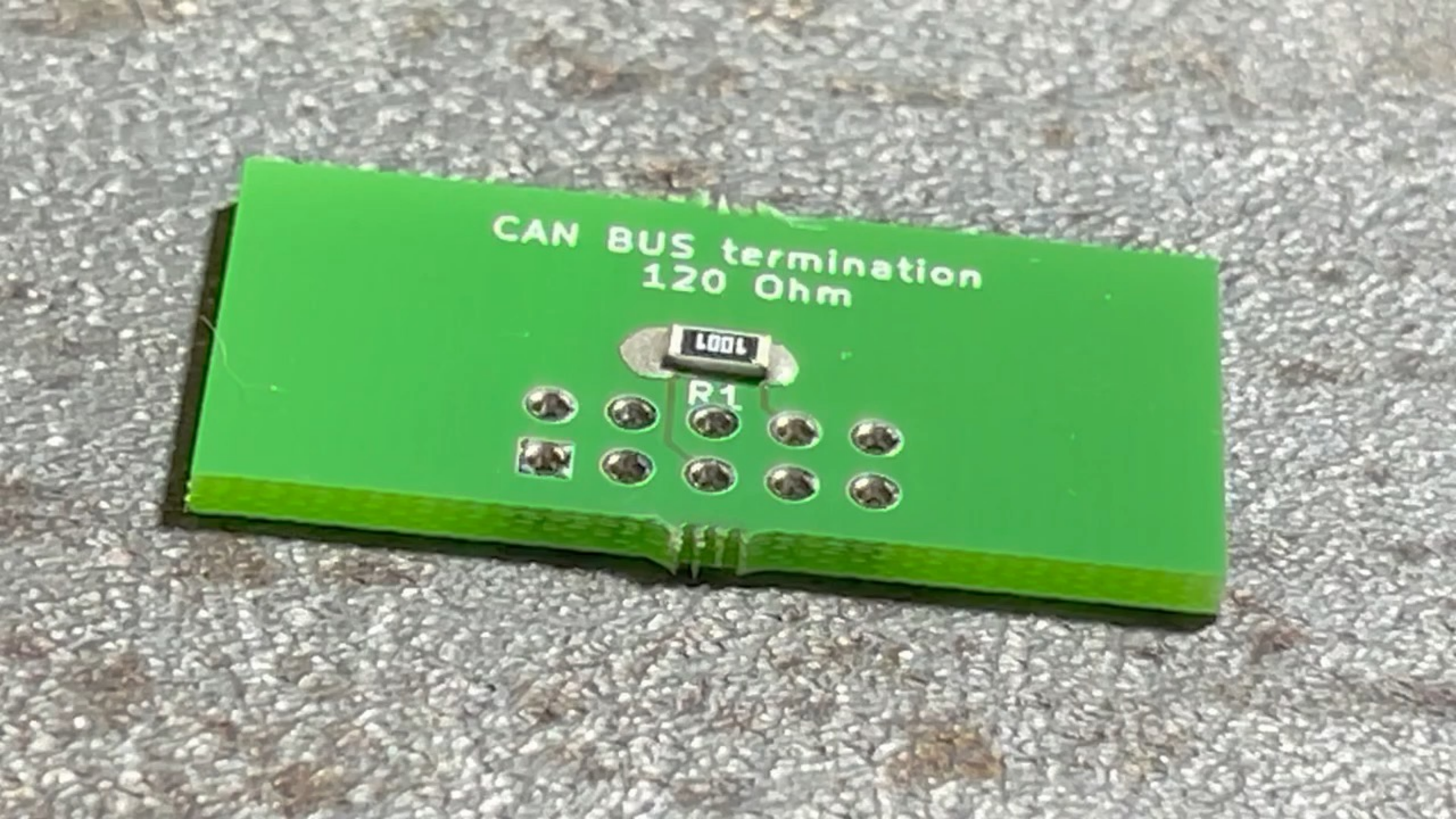




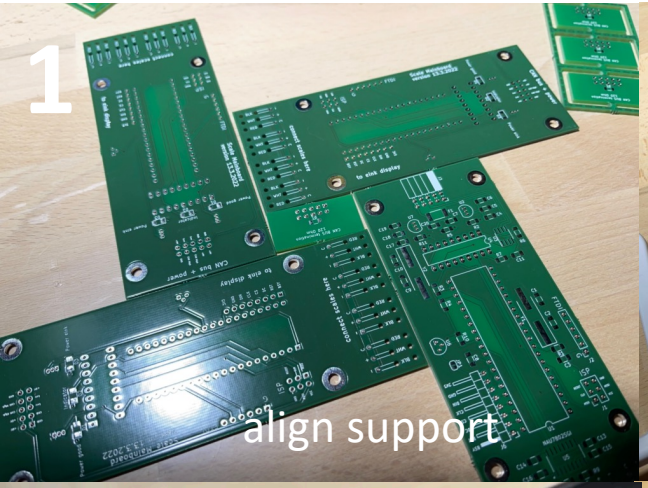


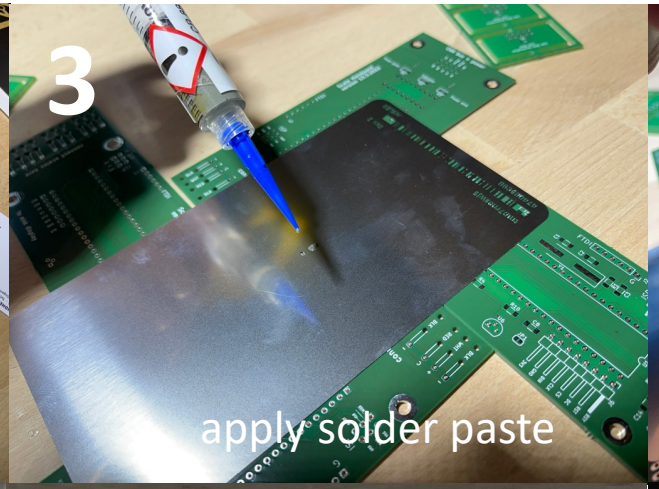
Reflow

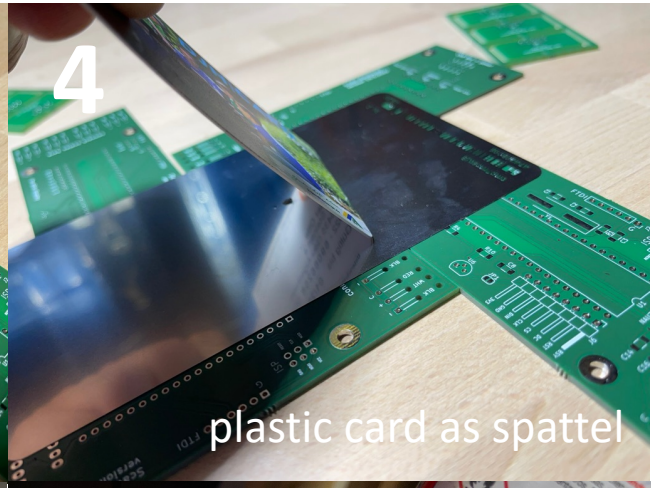
- Good how to: reflow by hand: https://www.sparkfun.com/tutorials/58

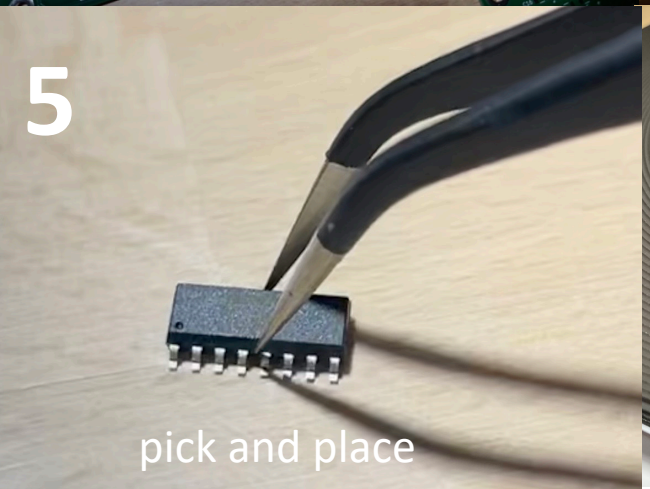

CAN BUS termination
120 Ohm
R1

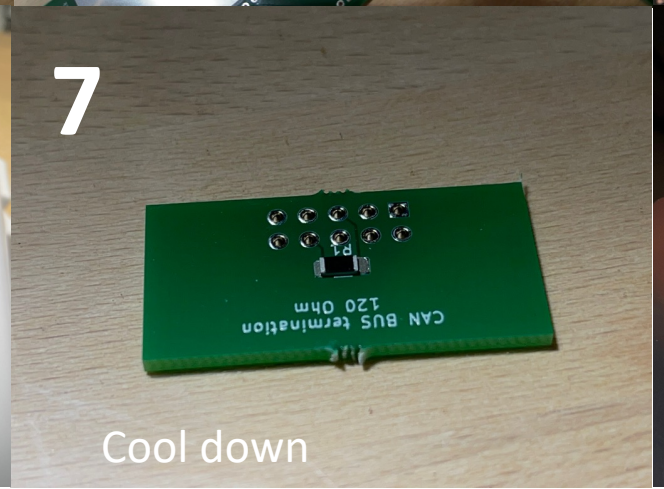
Reflow step by step

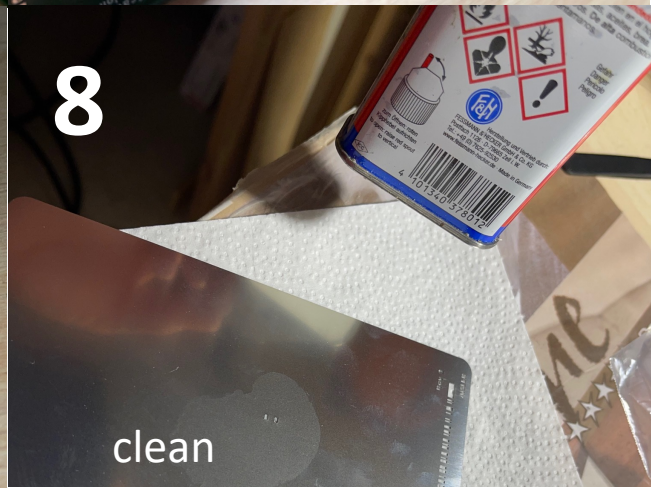




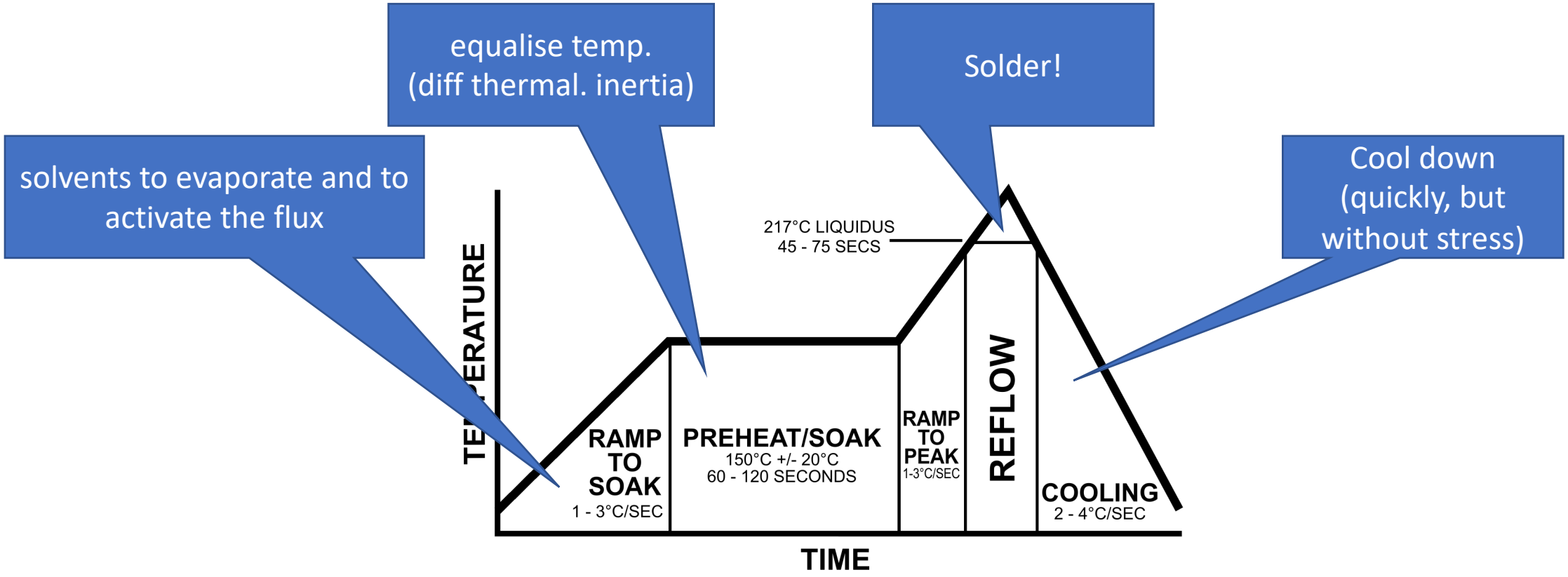




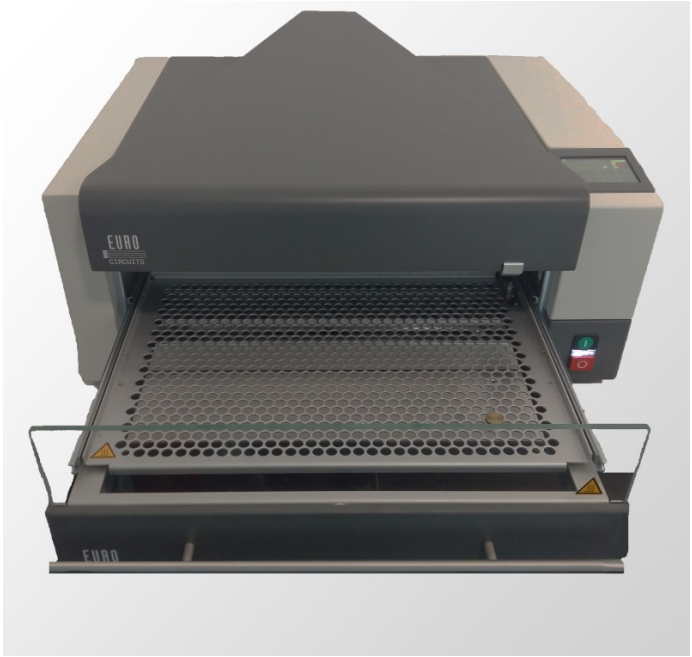




Reflow profile characteristics



Reflow Ovens



eC-reflow-mate V4 from eurocircuits

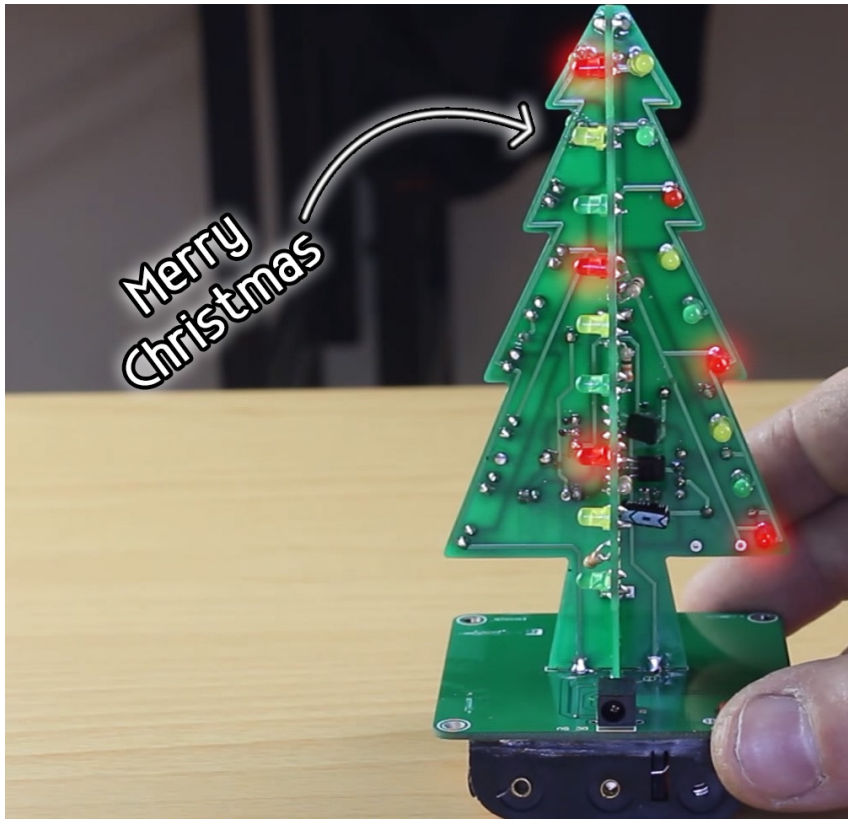
time dependent temperature

Conveyor belt

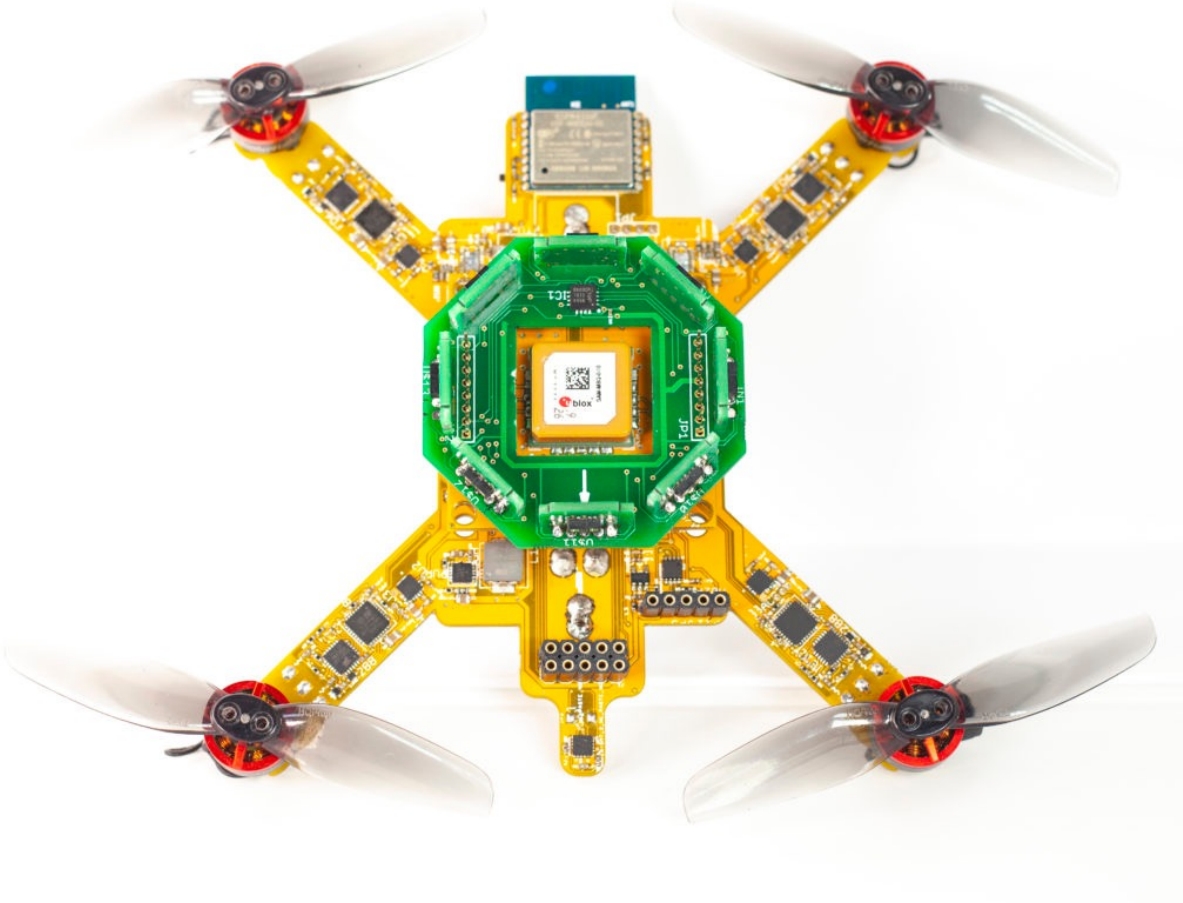
Fixed heating zones

Arbitrary PCB shapes and holes

PCB Christmas Tree
https://electronoobs.com/PCB_prototype17.php



ARDUBEE, http://luminousbe.es/ardubee/

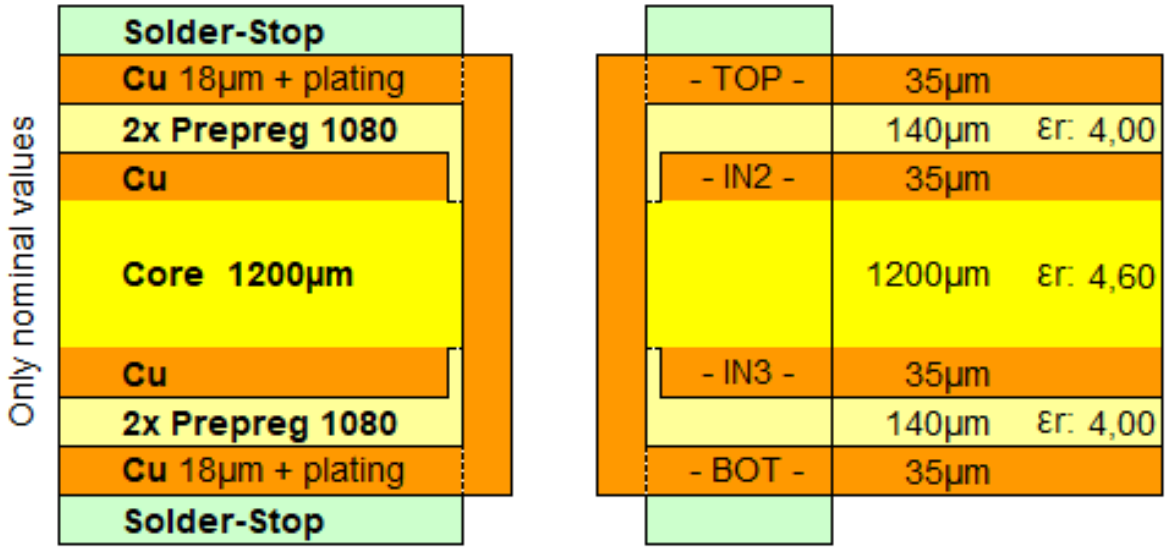


PCB properties

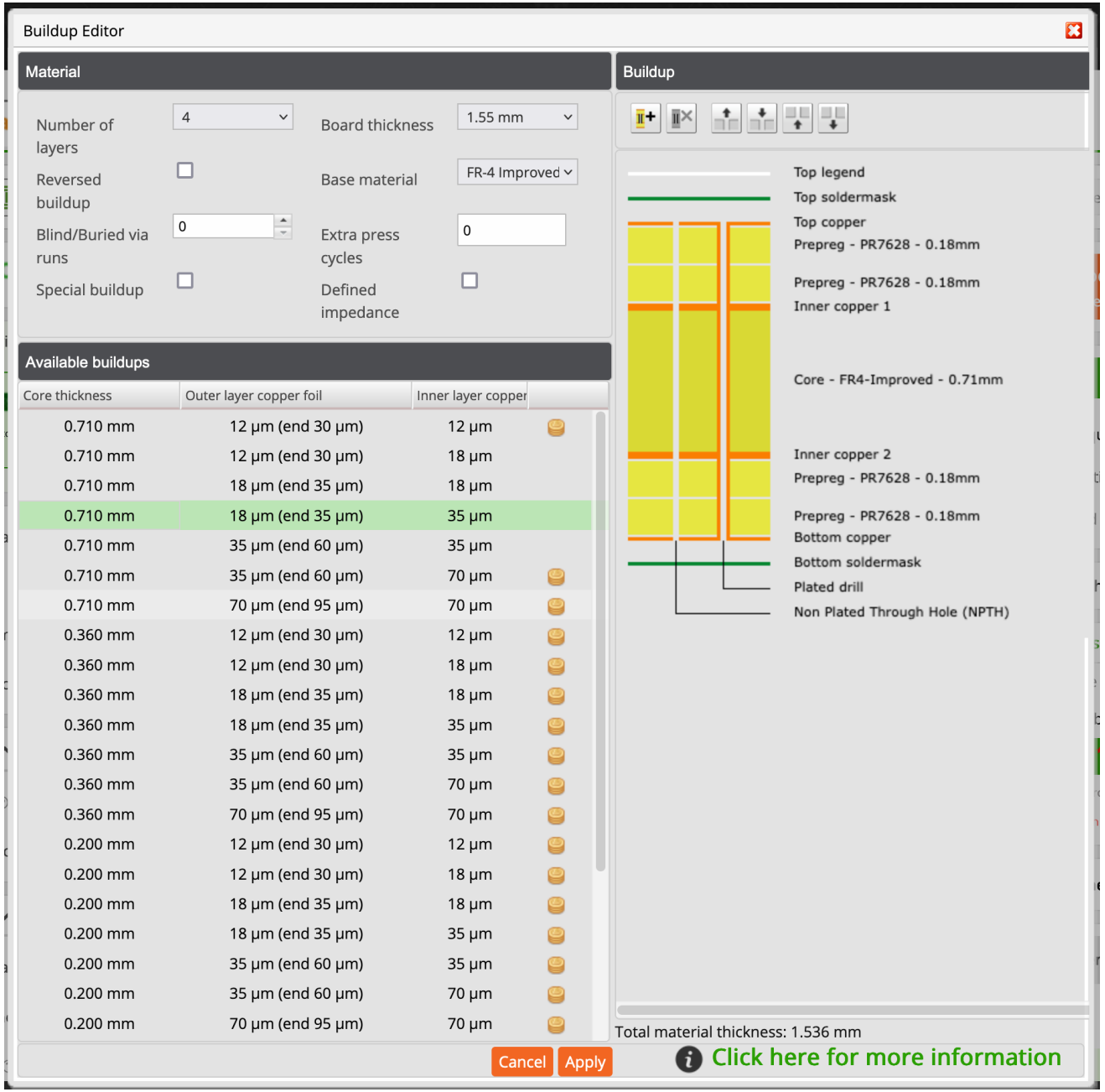
- Material: FR-4 (standard), aluminium, copper base, etc.
- Flexible, High frequency, high thermal loads, etc.
- Layer buildup for PCBs

multi-circuit-boards.eu

- (companies also offer assembly)

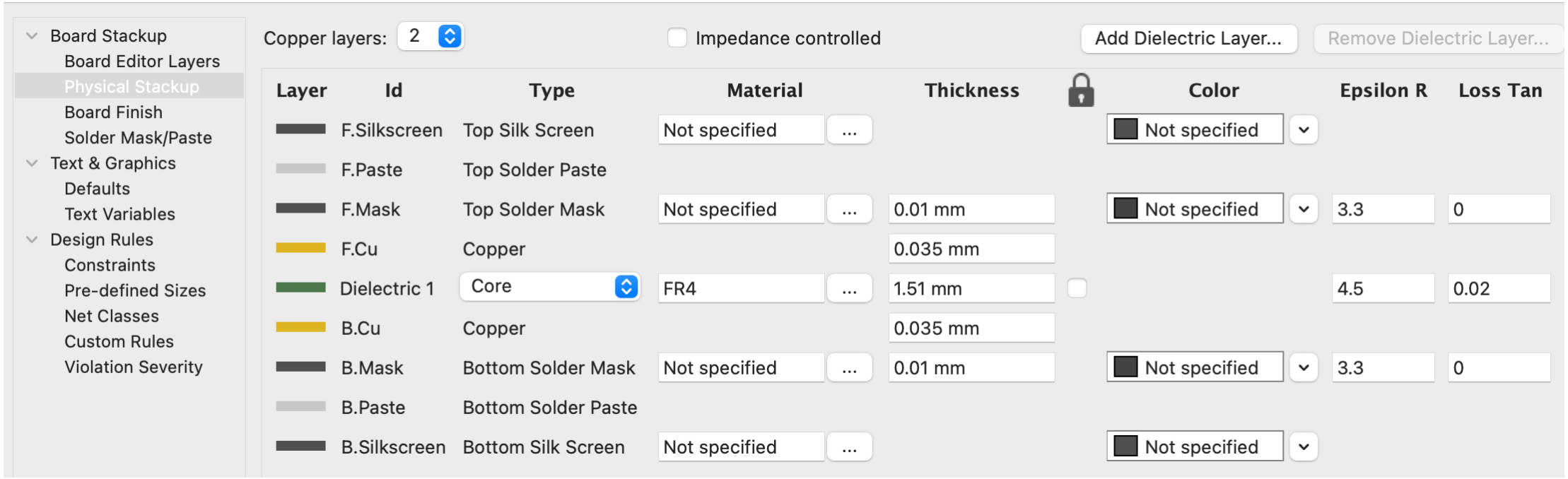


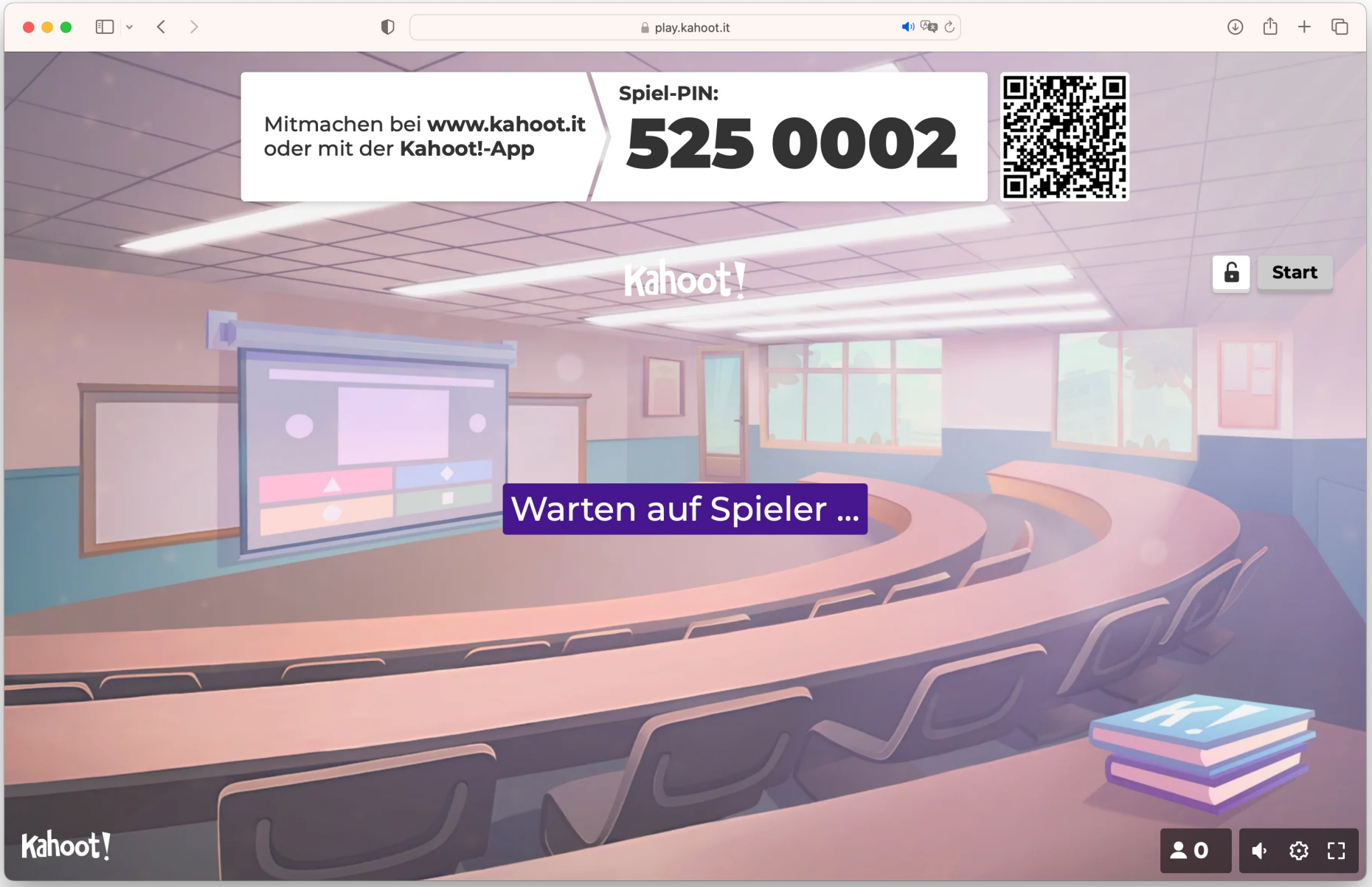
Example PCB build up from eurocircuits.com



Board Setup

PCB Editor: File > Board Setup ← match it with PCB manufacturer





https://create.kahoot.it/creator/f46b0b61-21d1-4e46-8356-bedb784061e2

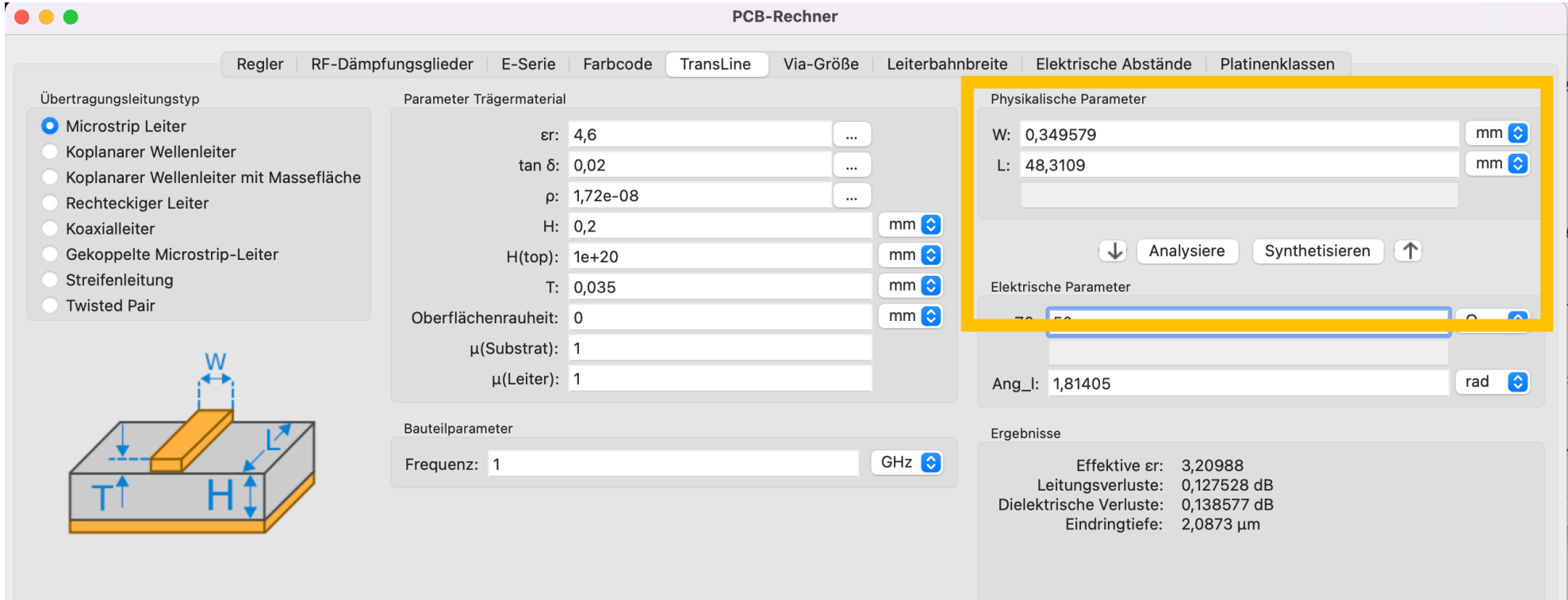
2) PCB design software overview

KiCAD import from other tools:

- EAGLE (successor: Fusion) ← KPH workshop standard
- Altium Circuit Maker/Studio / Designer
- CADSTAR

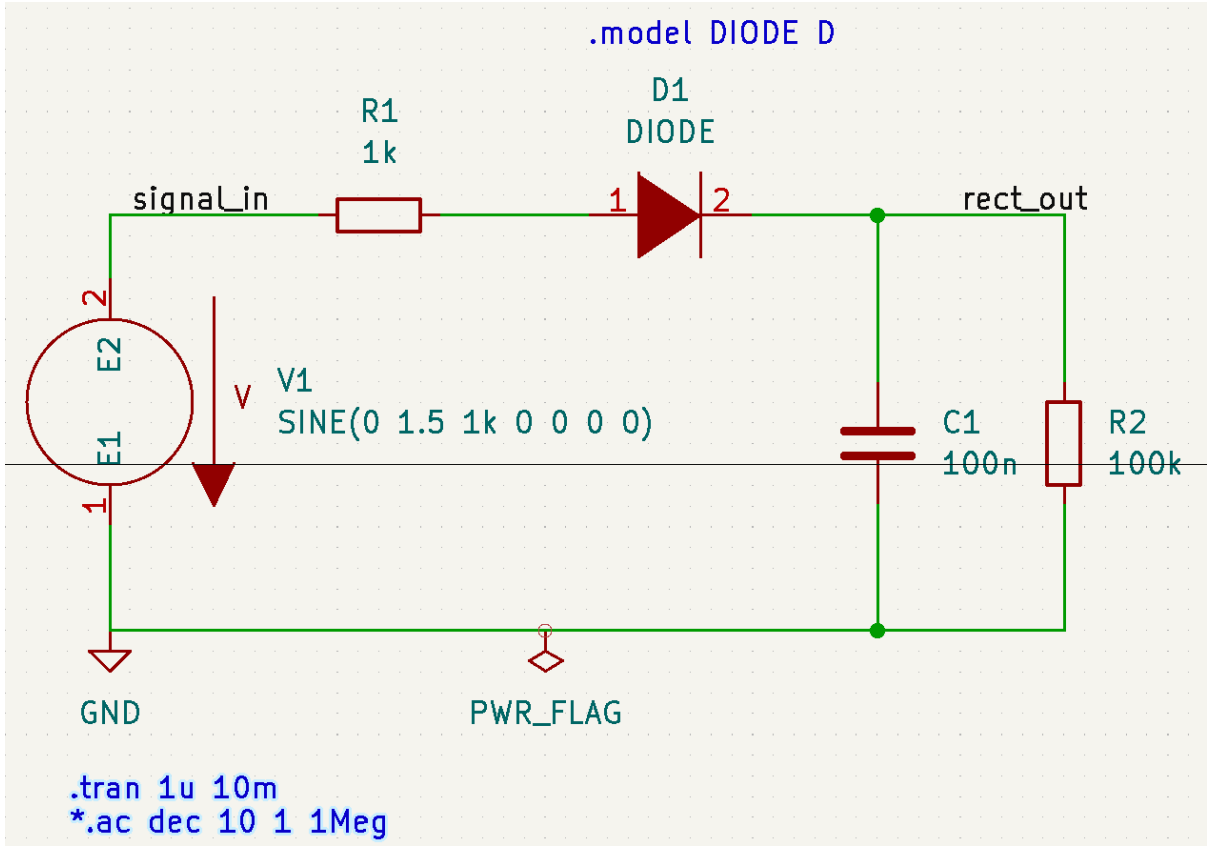
3) KiCAD’s advanced functionality showcase: impedance

- Calculate trace width: project window -> calculator tools.
- See PCB manufacturer for parameters
- Ground layer underneath
- Not needed for today’s slow control problem class



PSPICE simulation example

- See: “Pspace example”
- Run: “Inspect > Simulator”
  1. Hit ”play” button
  2. Add signals

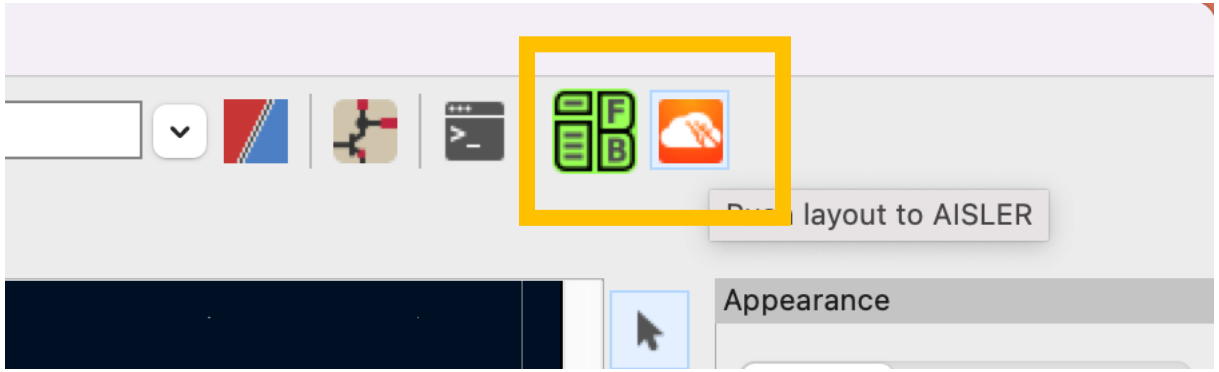


Hint on pin numbering:

- If inconsistent change via “Alternate node sequence” in "Properties” -> “Spice Model Editor”

Plugins (1/2): Aisler Push

- One click upload and PCB order
- Aisler = KiCAD platinum sponsor



Plugins (2/2): HTML BOM

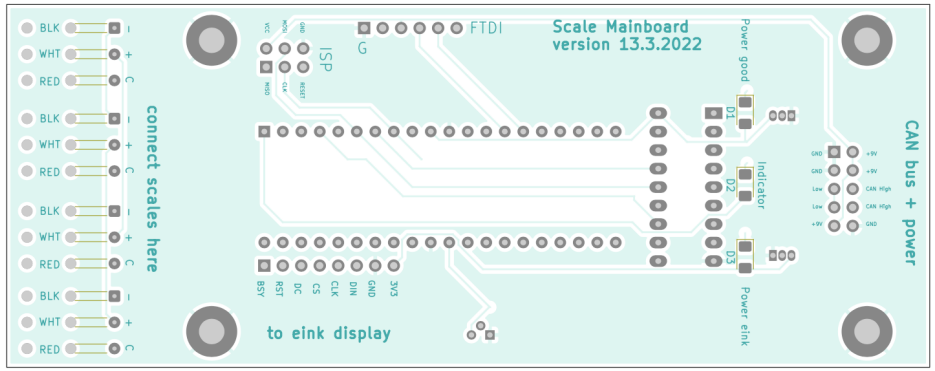
1284pTest2

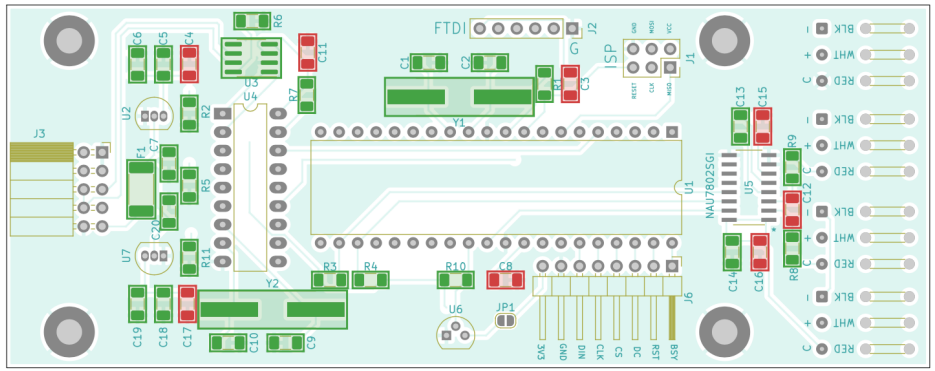
Rev:
2022-03-13 15:31:29

F FB B

Ref lookup | Filter

#	Sourced	Placed	Quantity	Value	References
1	☑	☐	8	100n	C3, C4, C8, C11, C12, C15, C16, C17
2	☑	☑	5	1µ	C6, C7, C14, C19, C20
3	☑	☑	4	22pF	C1, C2, C9, C10
4	☑	☑	2	10µ	C5, C18
5	☑	☑	1	330p	C13
6	☑	☑	3	120	R2, R5, R11
7	☑	☑	2	10k	R1, R7
8	☑	☑	2	2.2k	R3, R4
9	☑	☑	2	47	R8, R9
10	☑	☑	1	1k	R6
11	☑	☑	1	4.7k	R10
12	☐	☐	3	LED	D1, D2, D3
13	☐	☐	2	MCP1700-3302E_TO92	U2, U7
14	☐	☐	1	ATmega1284P-P	U1
15	☑	☑	1	SN65HVD230	U3
16	☐	☐	1	MCP2515	U4
17	☑	☐	1	NAU7802SGI	U5
18	☐	☐	1	DS18B20	U6
19	☑	☑	1	12MHz	Y1
20	☑	☑	1	8MHz	Y2
21	☑	☑	1	Fuse/Polyfuse 500mA	F1
22	☐	☐	1	SolderJumper_2_Open	JP1
23	☐	☐	4	Conn_01x03_Male	J4, J5, J7, J8
24	☐	☐	1	AVR-ISP-6	J1
25	☐	☐	1	Conn_01x06	J2
26	☐	☐	1	Conn_02x05_Odd_Even	J3
27	☐	☐	1	Conn_01x08	J6

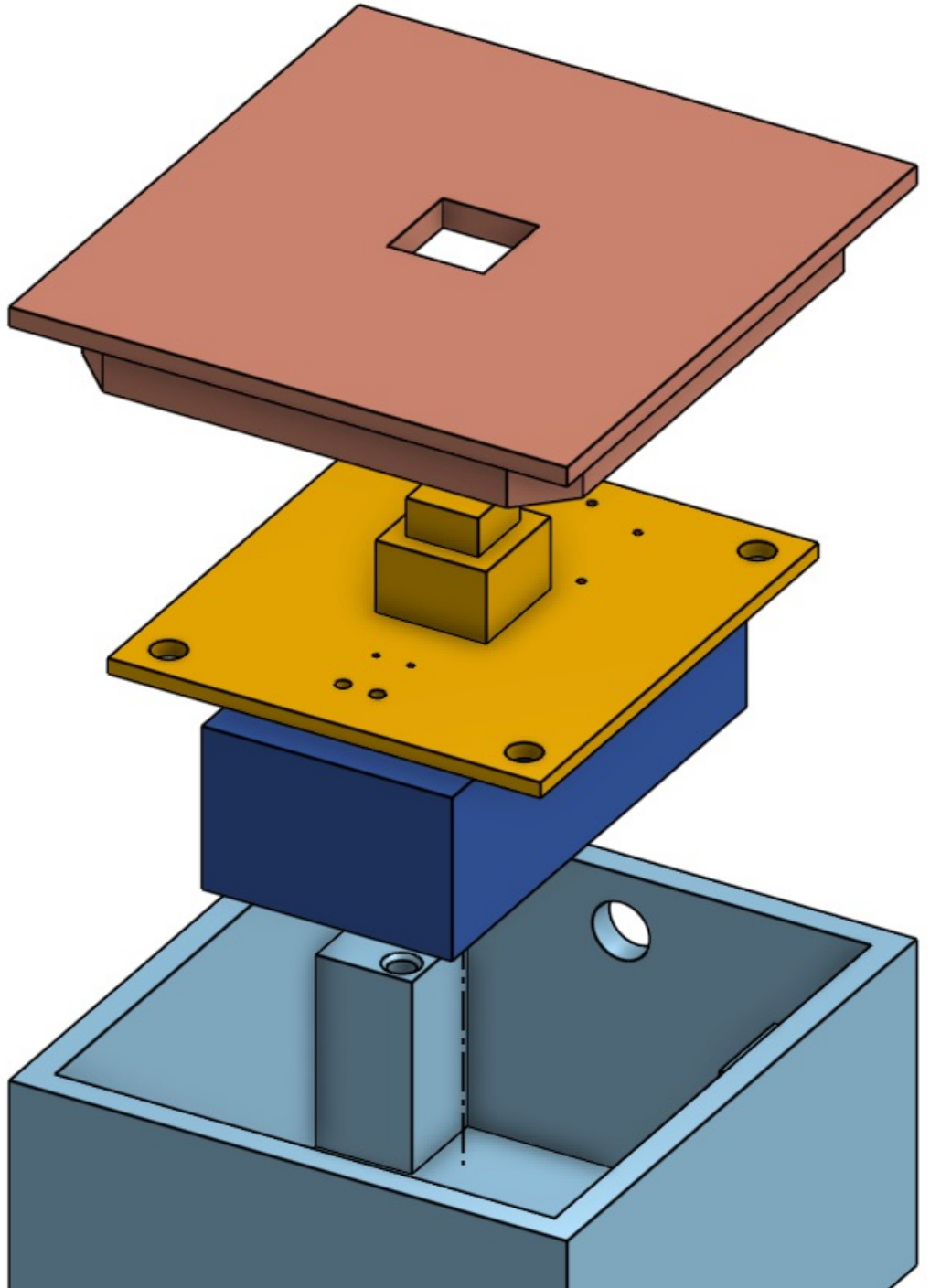




4) 1st project: Flash light

- (in cooperation with 3D design)

- Think of an amplifier in a box, but for today's problem class a bit simpler
- https://cad.onshape.com/documents/f3df22f41f5956c250e92d72/w/e39ec1ee5ab3727597d41460/e/9d99f21896da01cb25f775fc?renderMode=0&uiState=62a8a9478ac6385651dec014



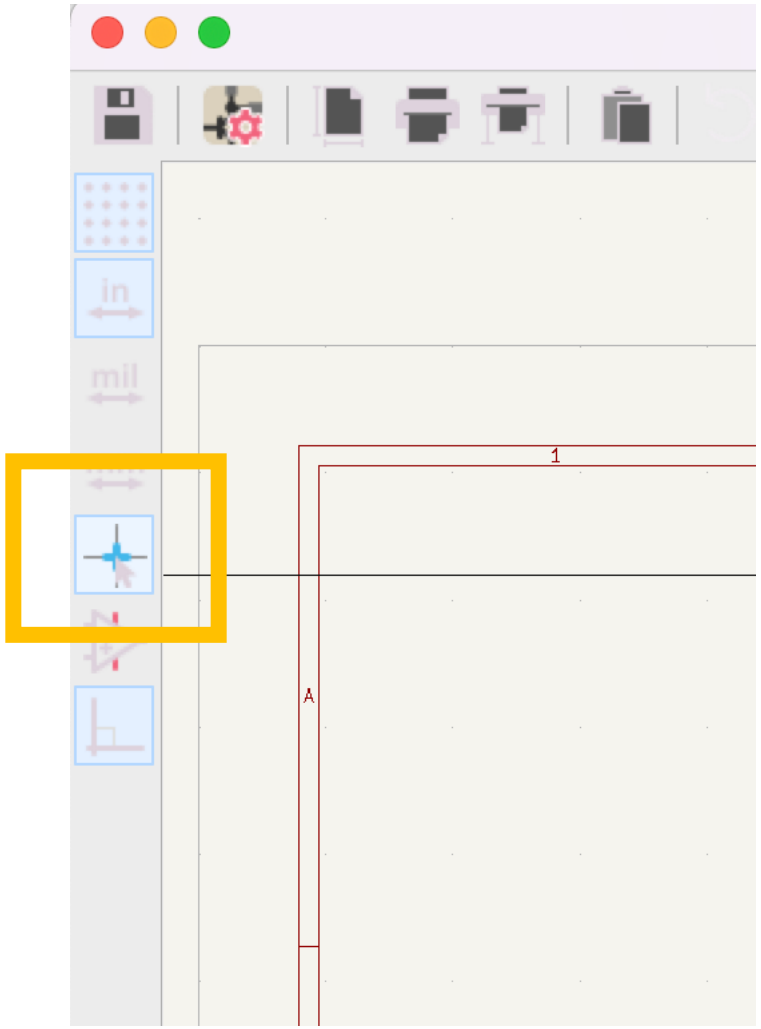
Since 2020: new procedure

Treat delivery dates > 1 week as infinite.

1. buy ALL parts
    1. Not avail? Check different package
    2. Alternative distributor?
    3. But reliable, eg via https://octopart.com
       otherwise fraud possible. Often: used or “relabelled” part (eg 5A MOSFET becomes 10A version simply by changing the label)
2. Start layout / PCB

Hints for preferences

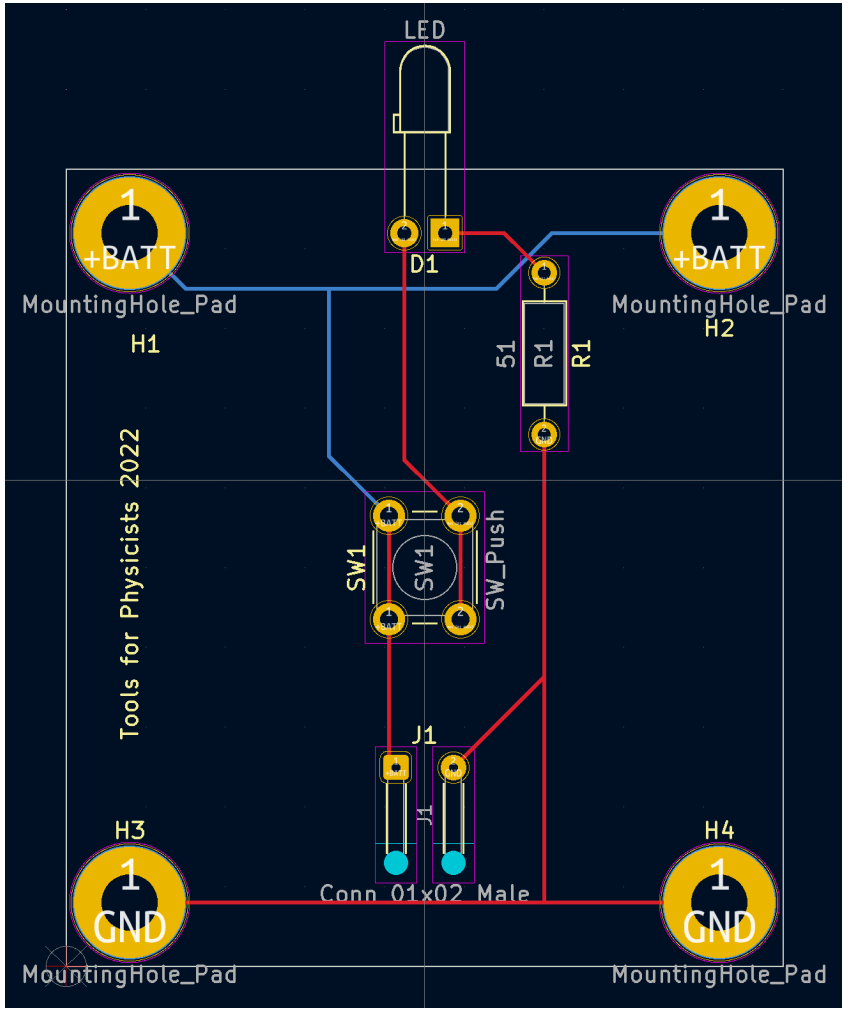
- Schematic-/PCB-Editor: “Always show crosshairs” for mouse pointer

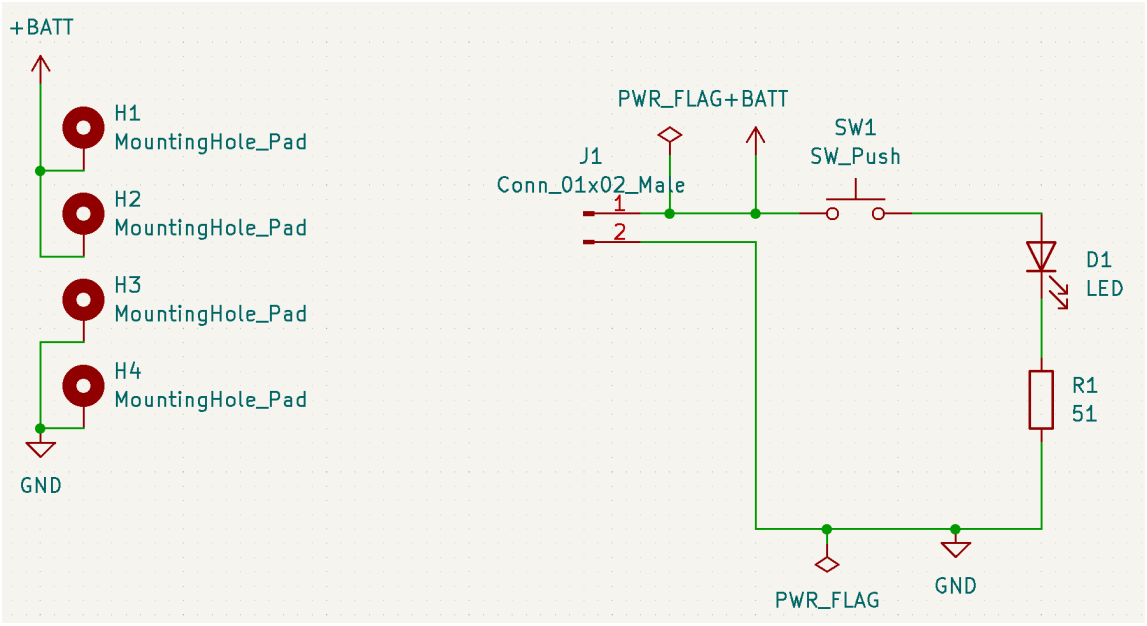


- PCB-Editor: enable “Show ratsnets with curved lines”

The design starts now!

- Together, step by step: basic concepts and workflow
- Alternatively: https://docs.kicad.org/7.0/en/getting_started_in_kicad/getting_started_in_kicad.html



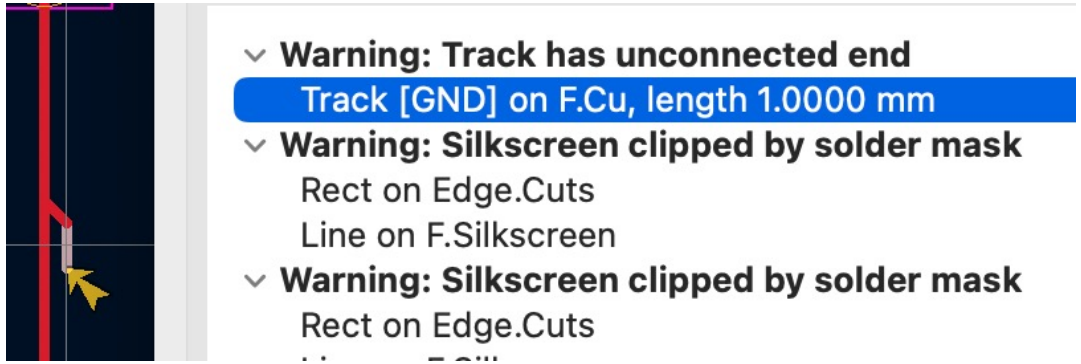


Symbol Fields Table

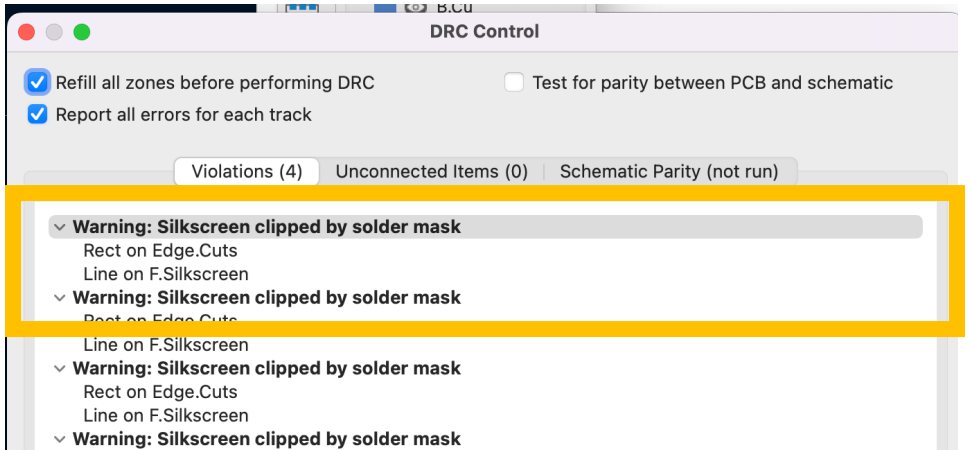
Reference	Value	Footprint	Datasheet	Qty
D1	LED	LED_THT:LED_D3.0mm_Horizontal_O6.35mm_Z2.0mm	~	1
> H1-H4	MountingHole_Pad	MountingHole:MountingHole_3.5mm_Pad	~	4
J1	Conn_01x02_Male	Connector_Wire:SolderWire-0.1sqmm_1x02_P3.6mm_D0.4mm_O	~	1
R1	51	Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Ho	~	1
SW1	SW_Push	Button_Switch_THT:SW_PUSH_6mm_H8mm	~	1

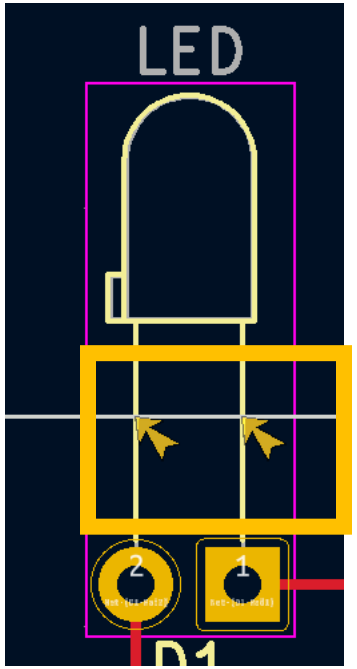
Typical DRC error messages and their solution

- Spot for hidden traces. Sometimes stubs are hidden behind other lines.



- Look out for the yellow arrows to locate DRC errors.





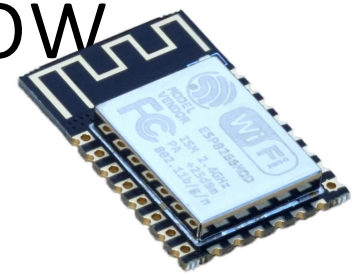
5) Custom Symbols and Footprints

- (Not for today's problem class)
- Detailed and up to date tutorial by the makes of KiCAD: https://docs.kicad.org/7.0/en/getting_started_in_kicad/getting_started_in_kicad.html

3 step process:

1. Create new symbols
2. Create new footprints
3. Linking symbols, footprints and 3D models

6) Bonus project: Wifi Lab environment Slow Control



- Submit with ESP 8266 data via WLAN SSID winulum to MQTT broker on campus
- Hints:
  - Make as many components equal, if possible (R 4,7 and R 10k-> R4,7)
  - Look into the datasheet of the components used what they need additionally.
  - Think of EM environment: Better smaller pull up/down than power savings.

Bauteil : Footprint Zuordnungen

1	C1 -	100 nF	Capacitor_SMD:C_0805_2012Metric
2	C2 -	100 nF	Capacitor_SMD:C_0805_2012Metric
3	C3 -	100 µF	Capacitor_Tantalum_SMD:CP_EIA-6032-28_Kemet-C
4	C4 -	10 µF	Capacitor_Tantalum_SMD:CP_EIA-3528-12_Kemet-T
5	C5 -	22 µF	Capacitor_Tantalum_SMD:CP_EIA-3216-12_Kemet-S
6	J3 -	USB_B_Micro	Connector_USB:USB_Micro-B_Wuerth_629105150521
7	J4 -	Conn_01x04_Male	Connector_PinHeader_2.54mm:PinHeader_1x04_P2.54mm_Vertical
8	R1 -	4,7k	Resistor_SMD:R_0805_2012Metric
9	R2 -	4,7k	Resistor_SMD:R_0805_2012Metric
10	R3 -	2.2k	Resistor_SMD:R_0805_2012Metric
11	R4 -	2.2k	Resistor_SMD:R_0805_2012Metric
12	R5 -	10k	Resistor_SMD:R_0805_2012Metric
13	R6 -	10k	Resistor_SMD:R_0805_2012Metric
14	R7 -	10k	Resistor_SMD:R_0805_2012Metric
15	SW1 -	SW_reset	Button_Switch_THT:SW_PUSH_6mm_H5mm
16	SW2 -	SW_flash	Button_Switch_THT:SW_PUSH_6mm_H5mm
17	U1 -	BME280	Package_LGA:Bosch_LGA-8_2.5x2.5mm_P0.65mm_ClockwisePinNumbering
18	U2 -	ESP-12F	RF_Module:ESP-12E
19	U3 -	AMS1117-3.3	Package_TO_SOT_SMD:SOT-223-3_TabPin2