



PCB Design with KiCad

Introduction and Guidance
for Physicists at HIM

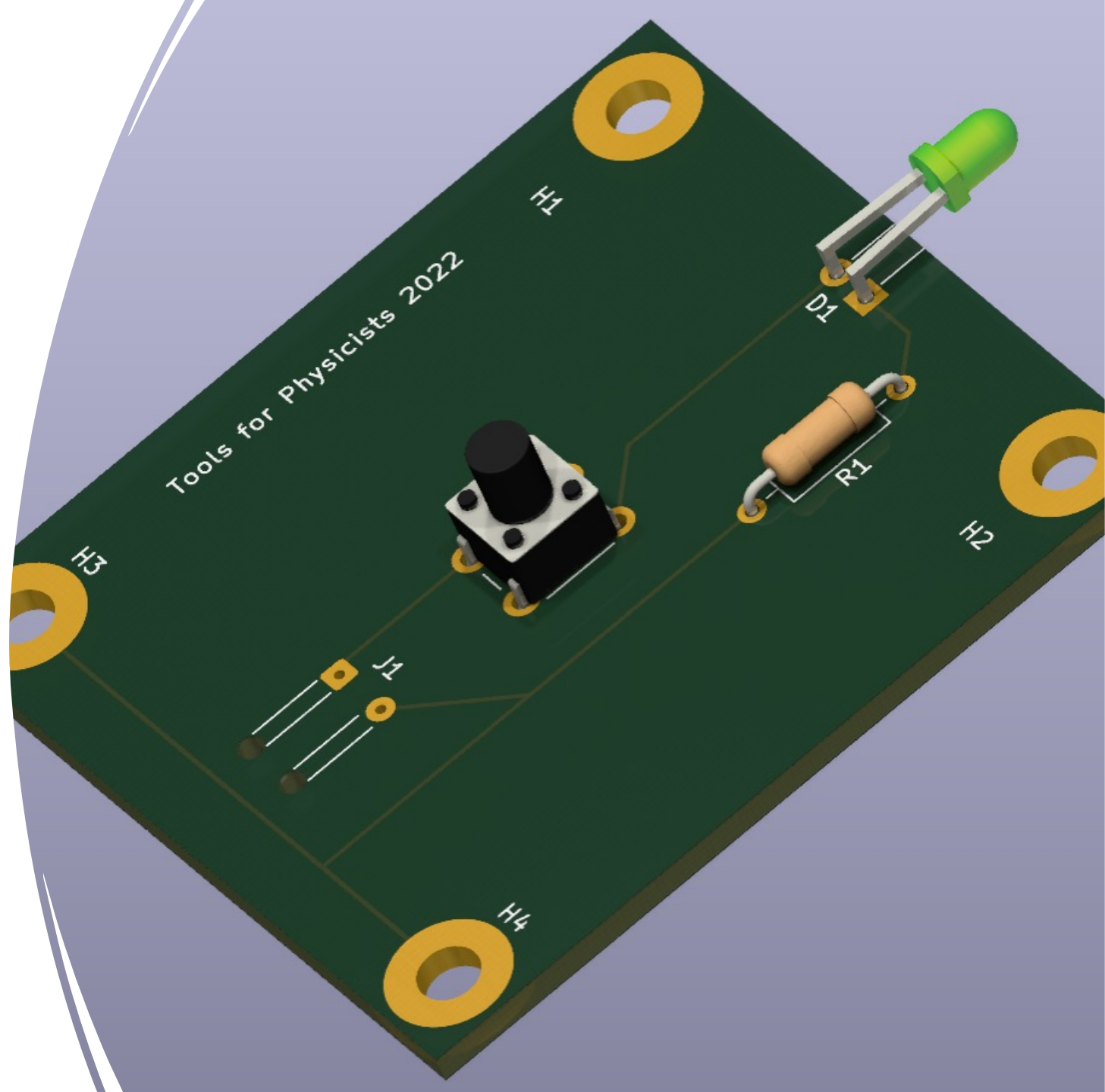
14.6.2022 – Peter-Bernd Otte - HI-Mainz

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 ...

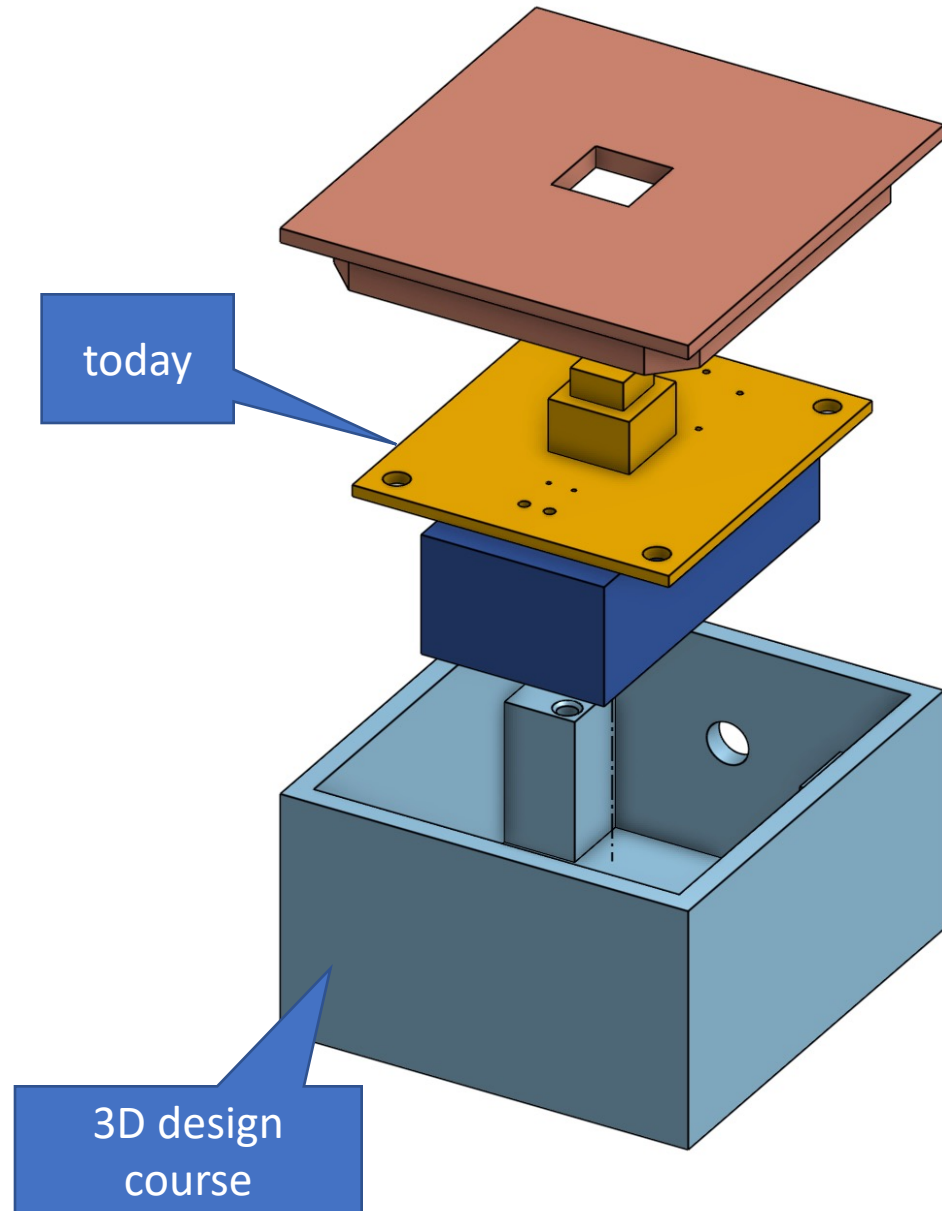
- high currents >1A → hot traces or contacts
- high voltages >42V → induce ventricular fibrillation
- high frequencies >20 MHz → impedance becomes important
- multi layer PCB >4 layers → complicated
- small / complicated parts → hard to solder
- large number of boards → usage of panels (dt. "Nutzen")

→ contact the electronics workshop at KPH (Dr. Werner Lauth and crew)

Focus today: slow control

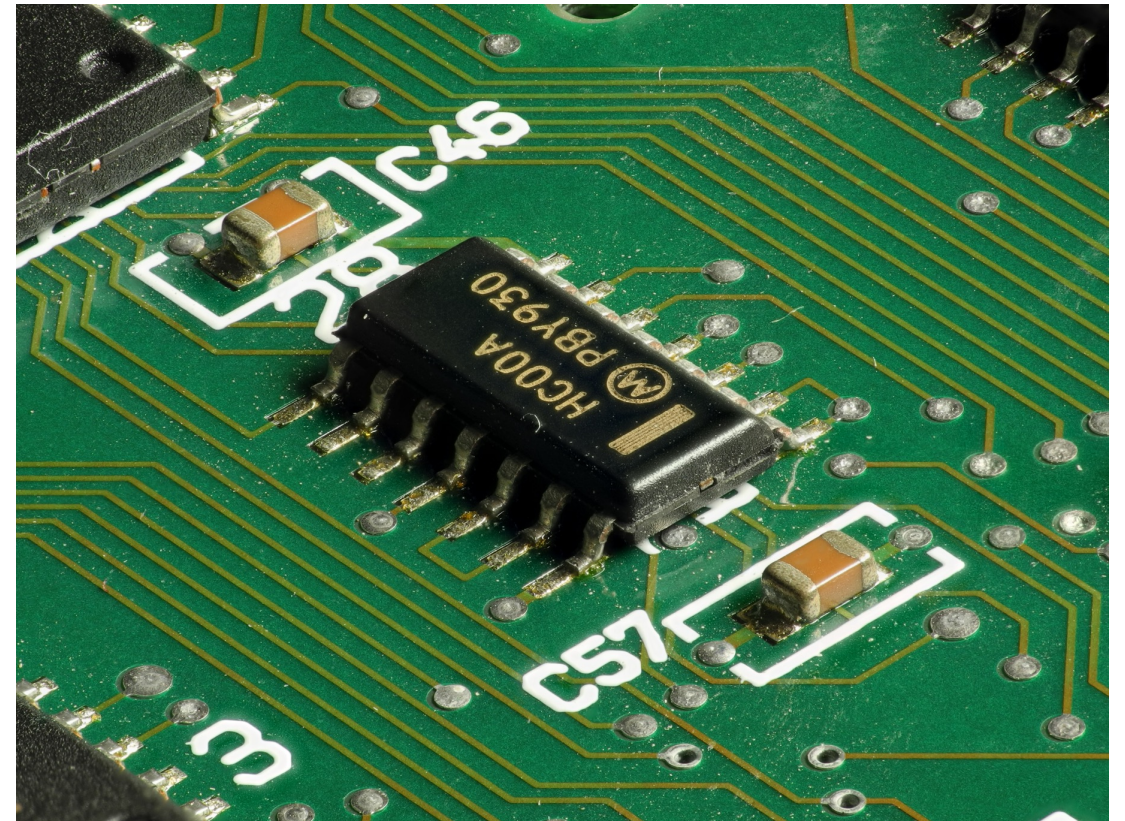
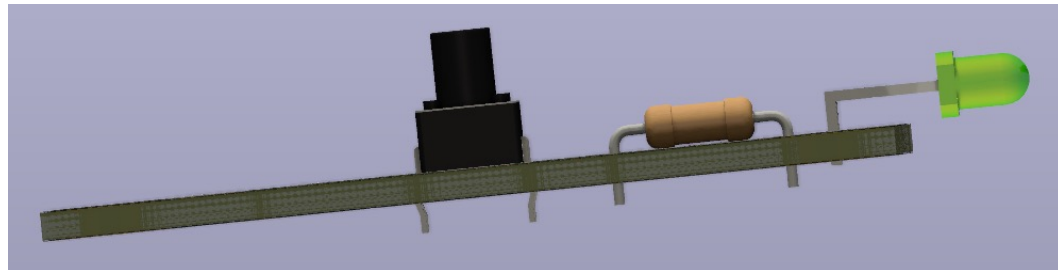
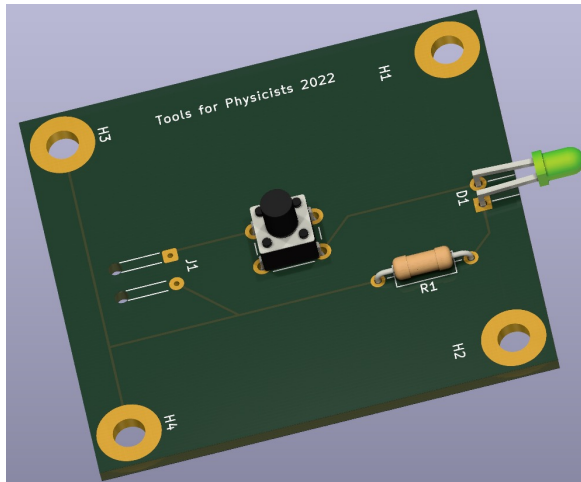
Today's overview

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

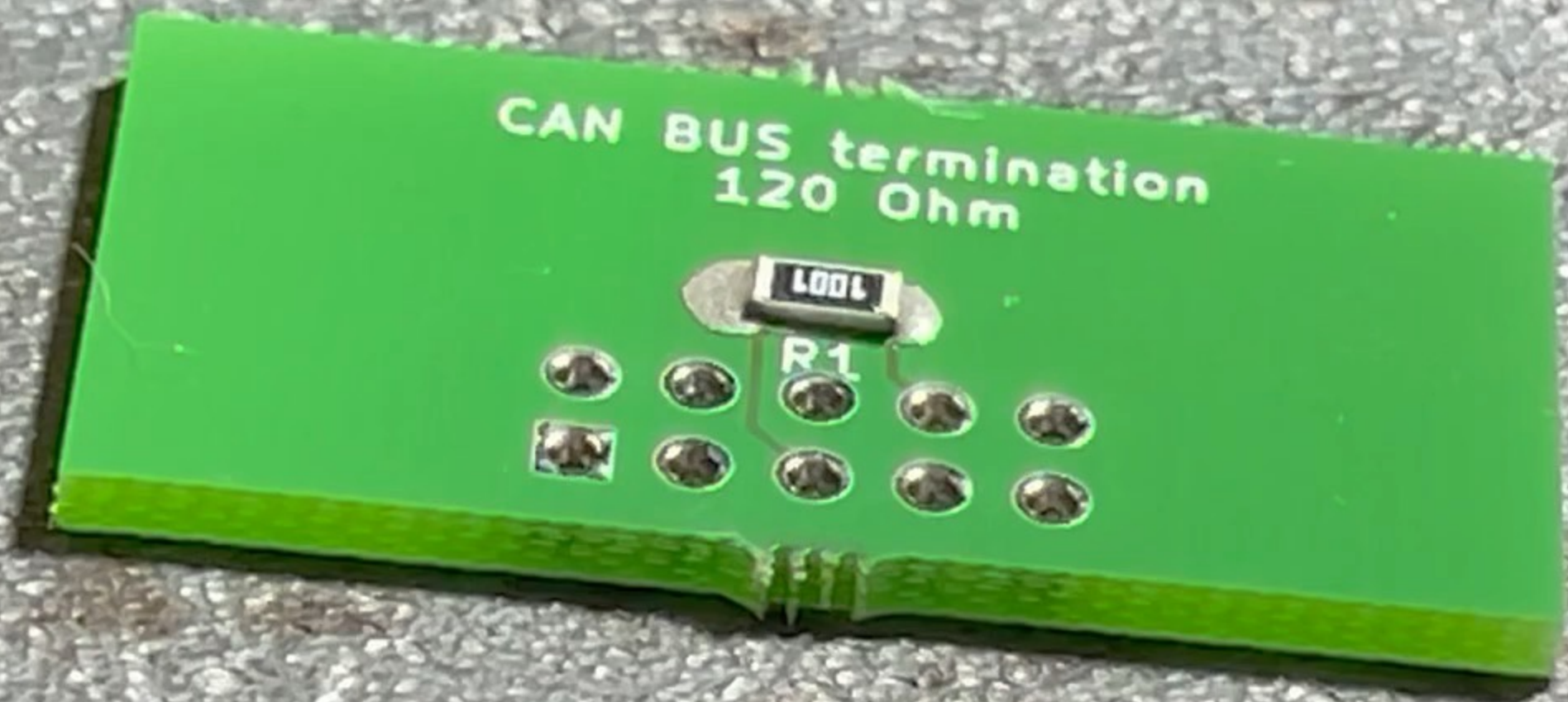
- Through hole technology (THT) and Surface Mount Device (SMD)



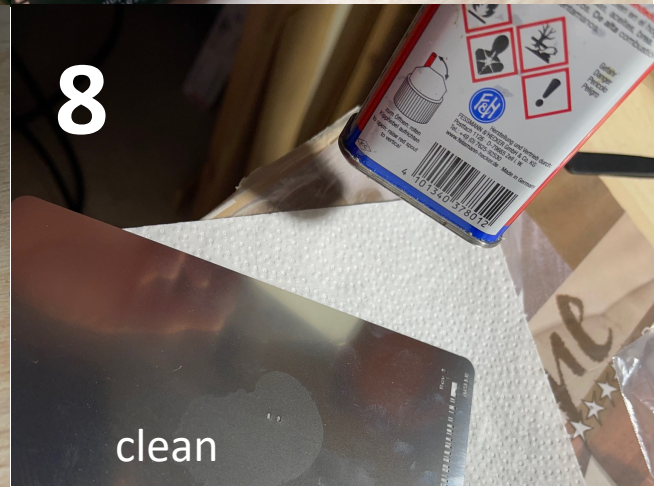
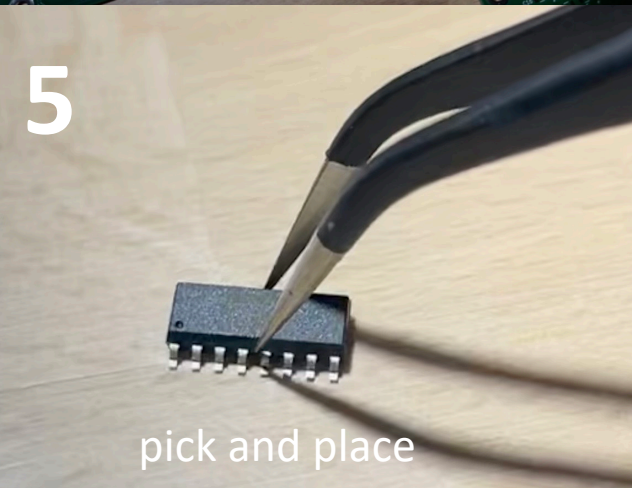
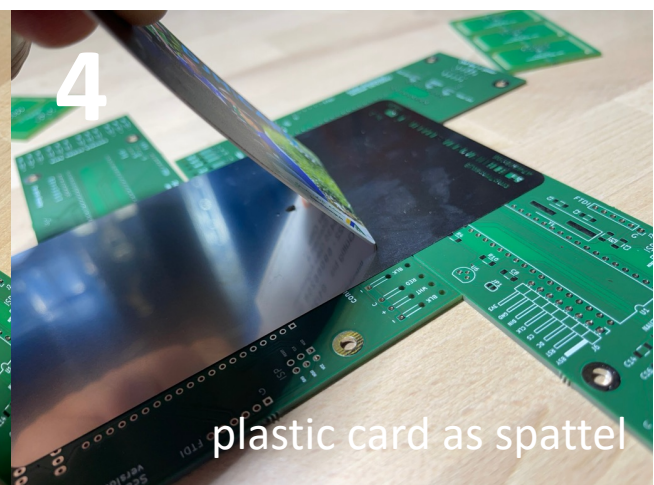
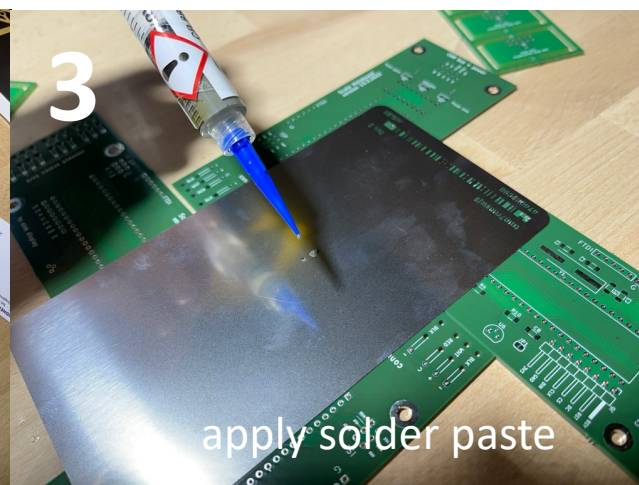
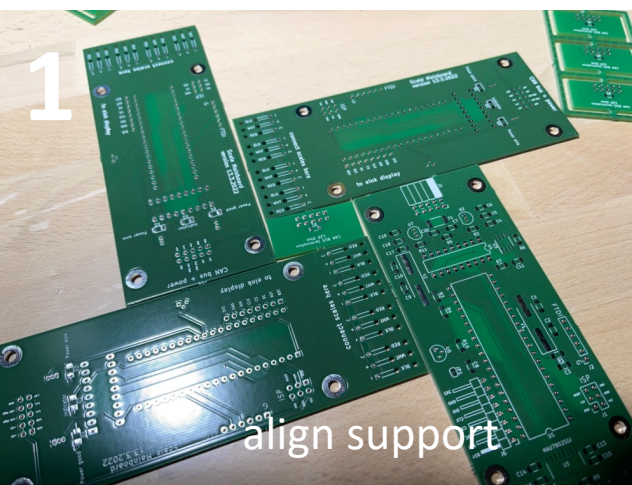
CAN BUS termination
120 Ohm

100Ω

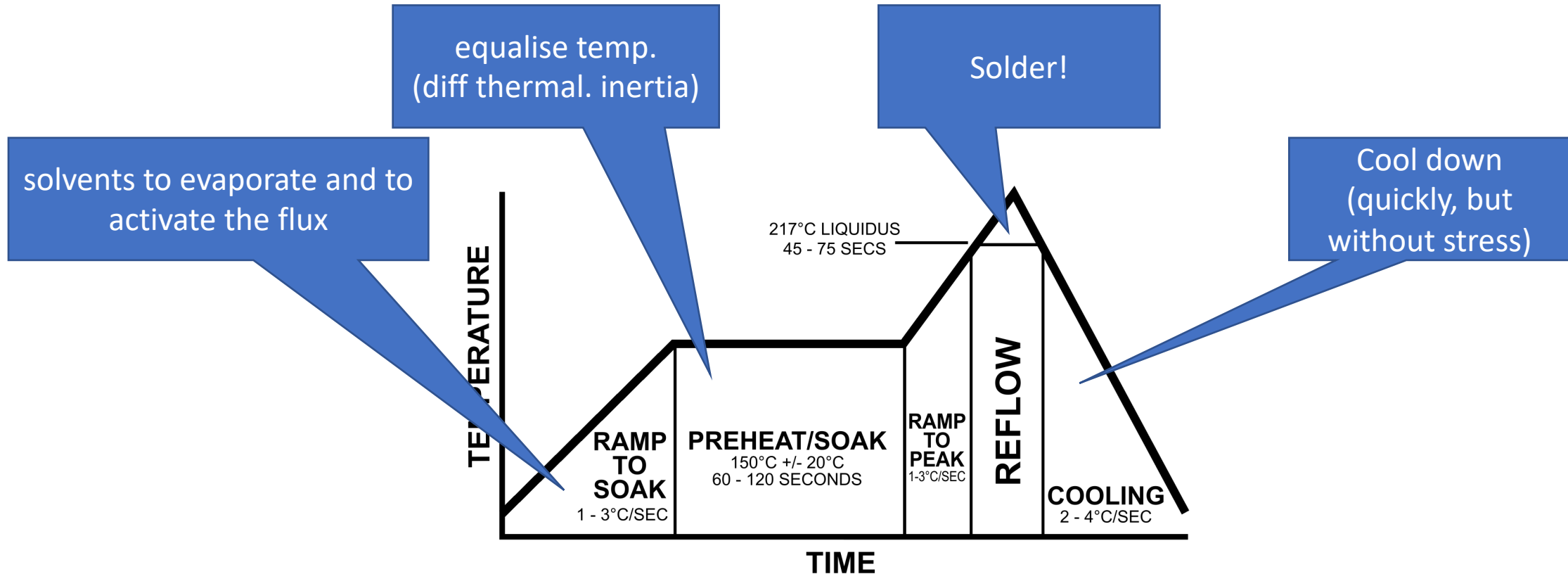
R1



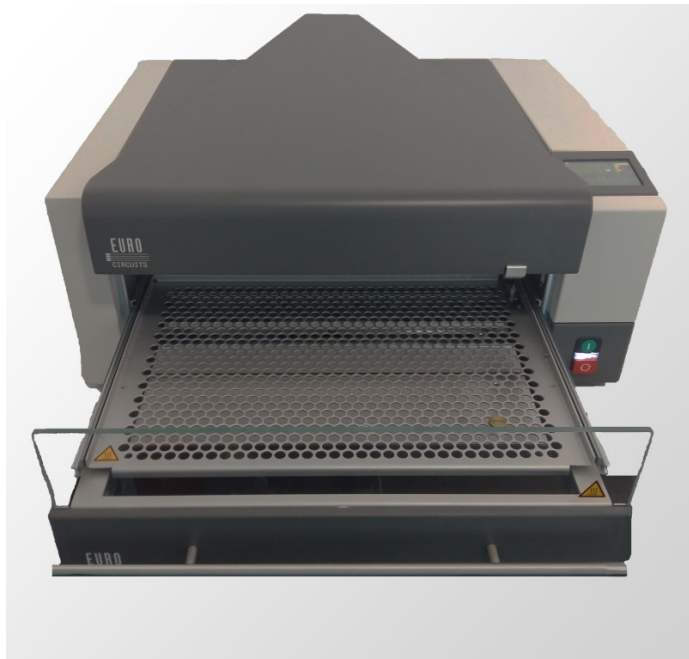
Reflow step by step



Reflow profile characteristics



Reflow Ovens



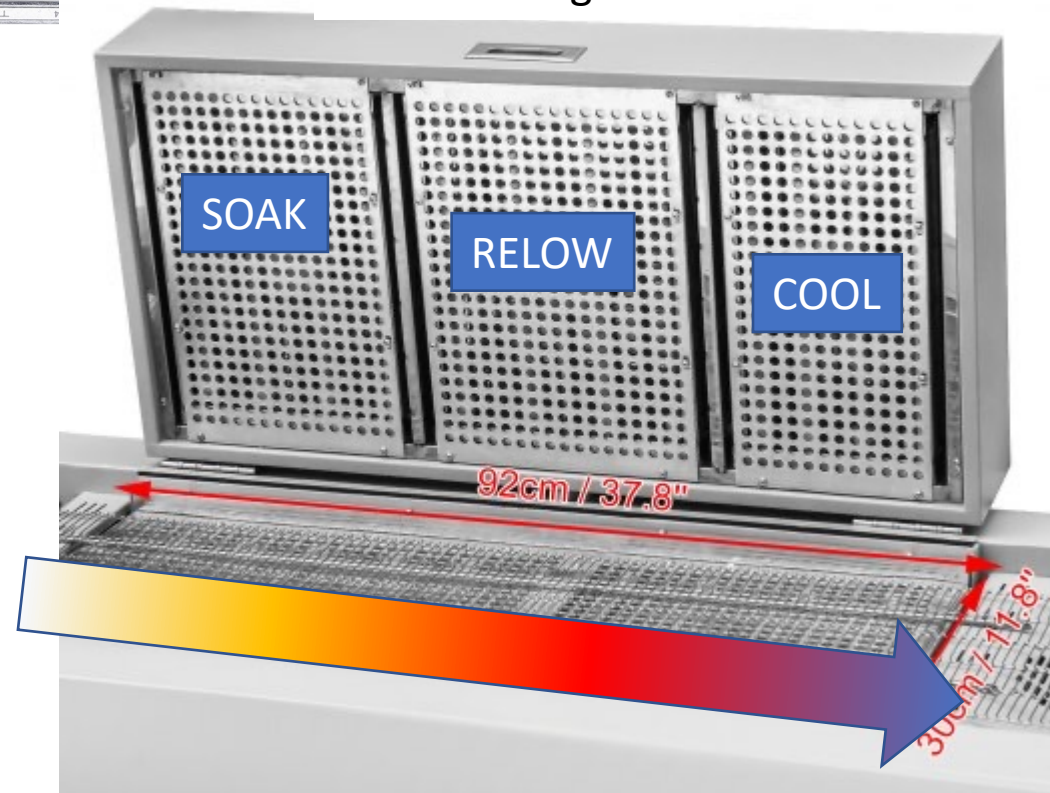
eC-reflow-mate V4 from eurocircuits

time dependent temperature



Conveyor belt

Fixed heating zones



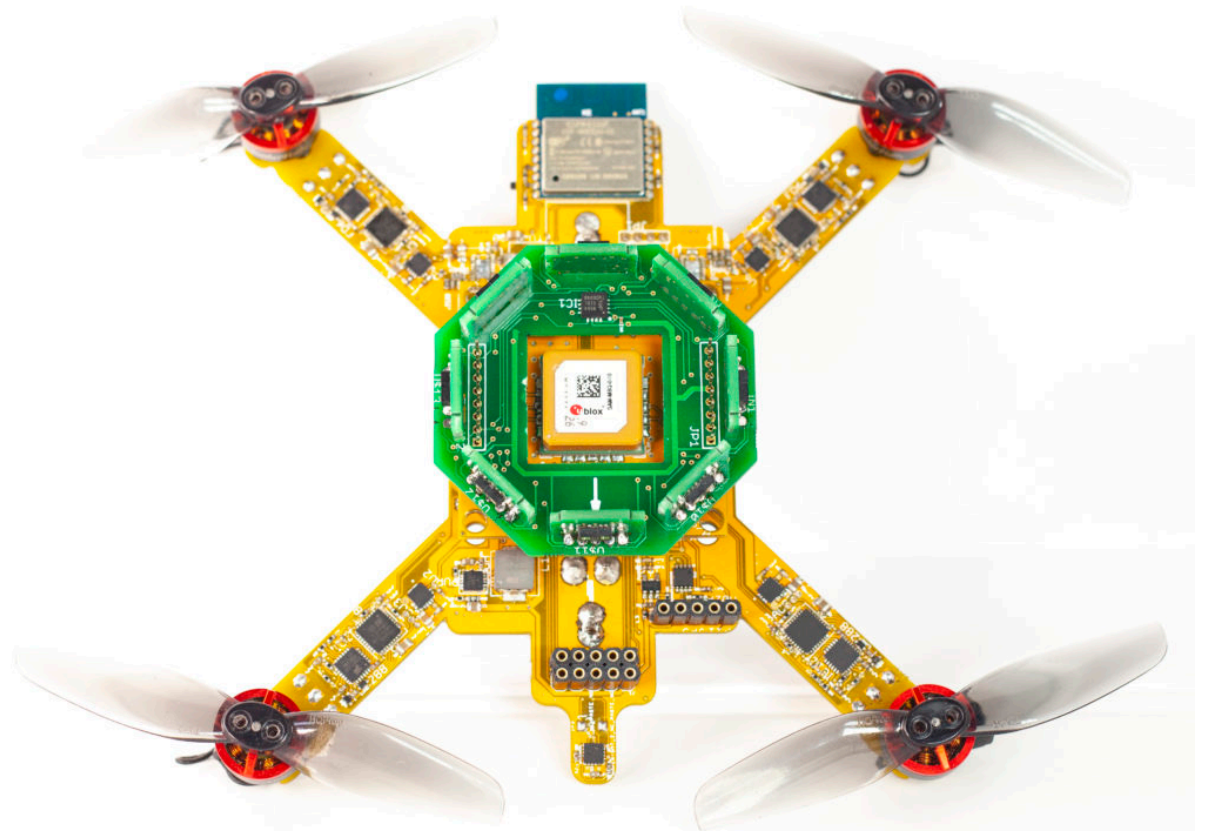
Arbitrary PCB shapes and holes

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)

Only nominal values

Solder-Stop
Cu 18µm + plating
2x Prepreg 1080
Cu
Core 1200µm
Cu
2x Prepreg 1080
Cu 18µm + plating
Solder-Stop

- TOP -	35µm	
	140µm	εr: 4,00
- IN2 -	35µm	
	1200µm	εr: 4,60
- IN3 -	35µm	
	140µm	εr: 4,00
- BOT -	35µm	

Example PCB build up from eurocircuits.com

Buildup Editor

Material

Number of layers: 4 Board thickness: 1.55 mm

Reversed buildup: Base material: FR-4 Improved

Blind/Buried via runs: 0 Extra press cycles: 0

Special buildup: Defined impedance:

Available buildups

Core thickness	Outer layer copper foil	Inner layer copper	
0.710 mm	12 µm (end 30 µm)	12 µm	☺
0.710 mm	12 µm (end 30 µm)	18 µm	
0.710 mm	18 µm (end 35 µm)	18 µm	
0.710 mm	18 µm (end 35 µm)	35 µm	
0.710 mm	35 µm (end 60 µm)	35 µm	
0.710 mm	35 µm (end 60 µm)	70 µm	☺
0.710 mm	70 µm (end 95 µm)	70 µm	☺
0.360 mm	12 µm (end 30 µm)	12 µm	☺
0.360 mm	12 µm (end 30 µm)	18 µm	☺
0.360 mm	18 µm (end 35 µm)	18 µm	☺
0.360 mm	18 µm (end 35 µm)	35 µm	☺
0.360 mm	35 µm (end 60 µm)	35 µm	☺
0.360 mm	35 µm (end 60 µm)	70 µm	☺
0.360 mm	70 µm (end 95 µm)	70 µm	☺
0.200 mm	12 µm (end 30 µm)	12 µm	☺
0.200 mm	12 µm (end 30 µm)	18 µm	☺
0.200 mm	18 µm (end 35 µm)	18 µm	☺
0.200 mm	18 µm (end 35 µm)	35 µm	☺
0.200 mm	35 µm (end 60 µm)	35 µm	☺
0.200 mm	35 µm (end 60 µm)	70 µm	☺
0.200 mm	70 µm (end 95 µm)	70 µm	☺

Buildup

Top legend

- Top soldermask
- Top copper
- Prepreg - PR7628 - 0.18mm
- Prepreg - PR7628 - 0.18mm
- Inner copper 1
- Core - FR4-Improved - 0.71mm
- Inner copper 2
- Prepreg - PR7628 - 0.18mm
- Bottom copper
- Bottom soldermask
- Plated drill
- Non Plated Through Hole (NPTH)

Total material thickness: 1.536 mm

Cancel Apply [Click here for more information](#)

Board Setup

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

The screenshot shows the 'Board Setup' dialog box in a PCB editor. On the left is a tree view with categories: Board Stackup, Board Editor Layers, Physical Stackup (selected), Board Finish, Solder Mask/Paste, Text & Graphics, Defaults, Text Variables, Design Rules, Constraints, Pre-defined Sizes, Net Classes, Custom Rules, and Violation Severity. The main area is titled 'Copper layers: 2' with a dropdown arrow. To the right of this are 'Impedance controlled' (unchecked) and two buttons: 'Add Dielectric Layer...' and 'Remove Dielectric Layer...'. Below is a table of layers with columns: Layer, Id, Type, Material, Thickness, a lock icon, Color, Epsilon R, and Loss Tan.

Layer	Id	Type	Material	Thickness	Lock	Color	Epsilon R	Loss Tan
	F.Silkscreen	Top Silk Screen	Not specified		<input checked="" type="checkbox"/>	Not specified		
	F.Paste	Top Solder Paste						
	F.Mask	Top Solder Mask	Not specified	0.01 mm		Not specified	3.3	0
	F.Cu	Copper		0.035 mm				
	Dielectric 1	Core	FR4	1.51 mm	<input type="checkbox"/>		4.5	0.02
	B.Cu	Copper		0.035 mm				
	B.Mask	Bottom Solder Mask	Not specified	0.01 mm		Not specified	3.3	0
	B.Paste	Bottom Solder Paste						
	B.Silkscreen	Bottom Silk Screen	Not specified			Not specified		



Mitmachen bei www.kahoot.it
oder mit der **Kahoot!-App**

Spiel-PIN:

373 5072



 0

Kahoot!



Start

<https://play.kahoot.it/v2/lobby?quizId=ecf9af5f-629a-47b9-9902-bbbfaf36424d>

Warten auf Spieler ...

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

The screenshot displays the 'PCB-Rechner' (PCB Calculator) interface. The 'TransLine' tab is active, showing various input fields for calculating trace impedance. A yellow box highlights the 'Physikalische Parameter' (Physical Parameters) section, which includes:

- W: 0,349579 mm
- L: 48,3109 mm

Below this, there are buttons for 'Analyse' and 'Synthetisieren'. The 'Elektrische Parameter' (Electrical Parameters) section shows:

- Ang_l: 1,81405 rad

The 'Ergebnisse' (Results) section at the bottom right displays the following values:

- Effektive ϵ_r : 3,20988
- Leitungsverluste: 0,127528 dB
- Dielektrische Verluste: 0,138577 dB
- Eindringtiefe: 2,0873 μm

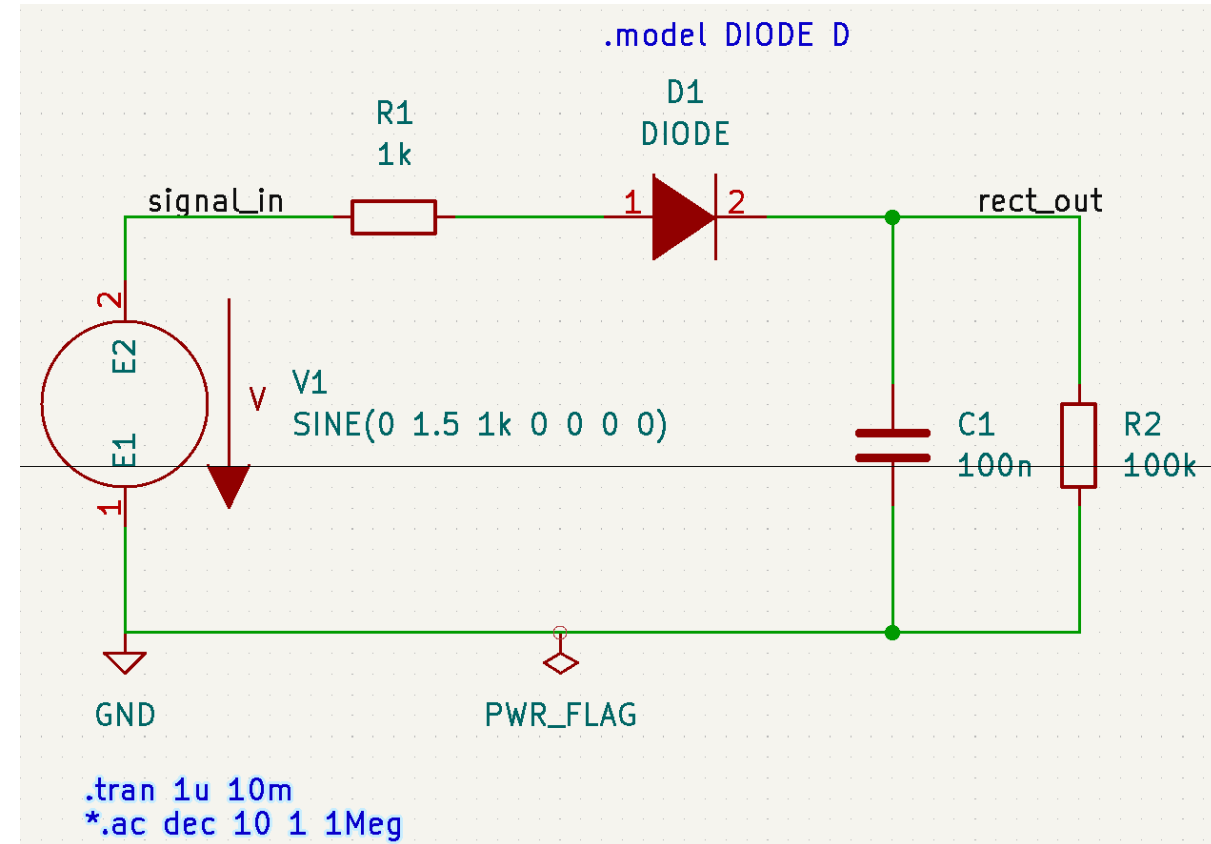
The interface also includes a 'Parameter Trägermaterial' (Substrate Parameters) section with fields for ϵ_r , $\tan \delta$, ρ , H, H(top), T, and surface roughness. A 3D diagram of a microstrip trace is shown at the bottom left, with labels for width (W), length (L), and height (H).

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”

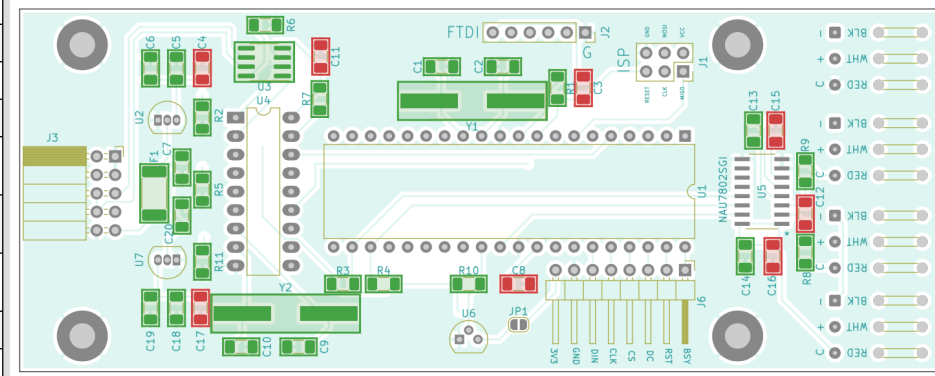
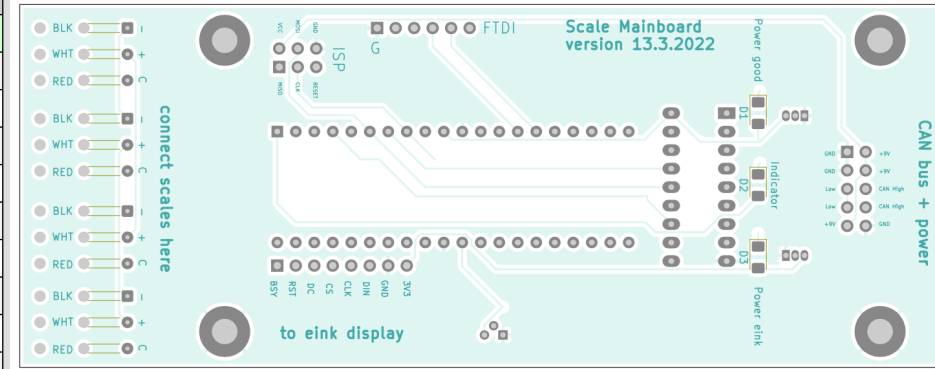


Plugins (1/2): Aisler Push

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



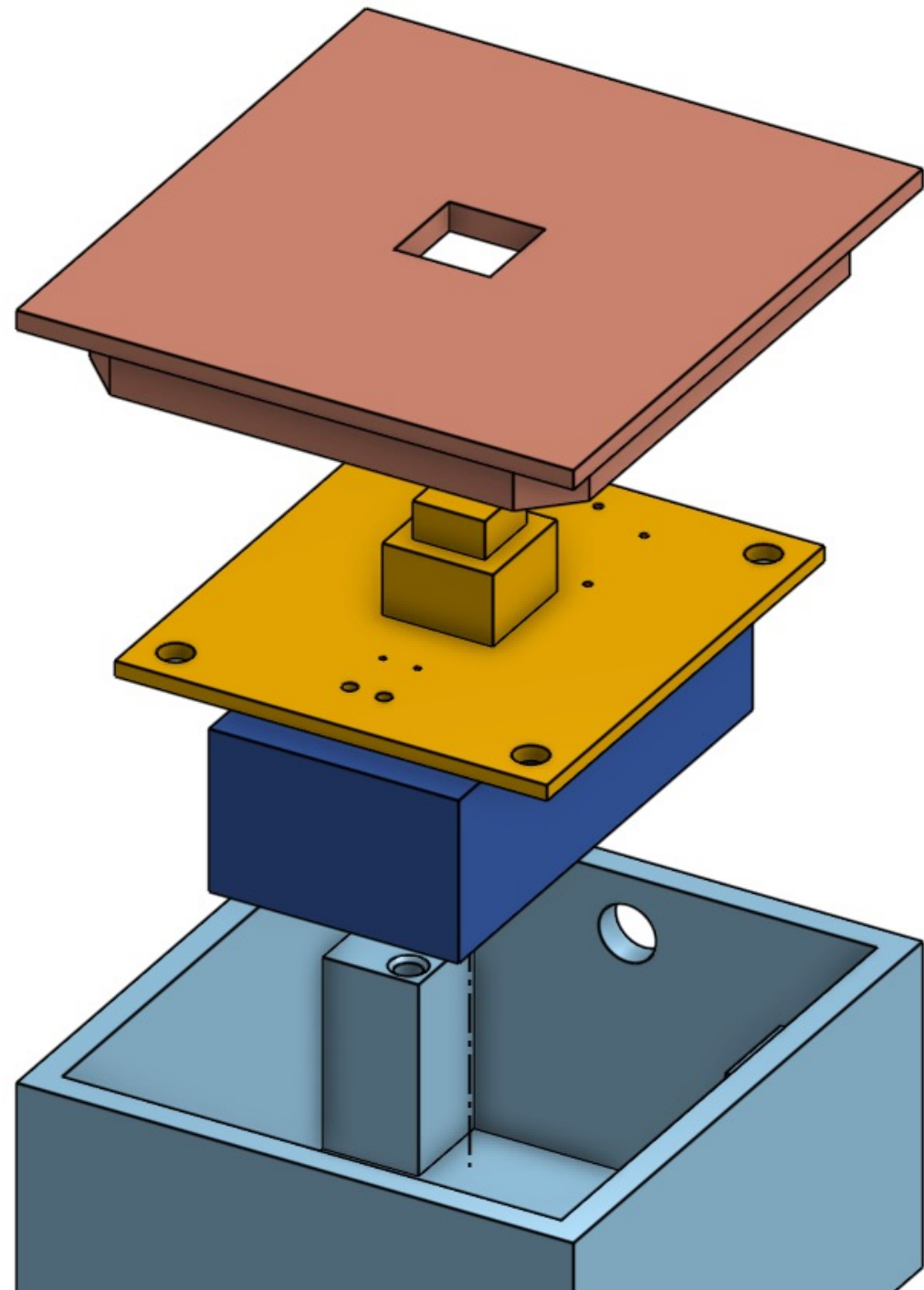
	Source	Placed	Quantity	Value	References
1	<input checked="" type="checkbox"/>	<input type="checkbox"/>	8	100n	C3, C4, C8, C11, C12, C15, C16, C17
2	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	5	1μ	C6, C7, C14, C19, C20
3	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	4	22pF	C1, C2, C9, C10
4	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	2	10μ	C5, C18
5	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1	330p	C13
6	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	3	120	R2, R5, R11
7	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	2	10k	R1, R7
8	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	2	2.2k	R3, R4
9	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	2	47	R8, R9
10	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1	1k	R6
11	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1	4.7k	R10
12	<input type="checkbox"/>	<input type="checkbox"/>	3	LED	D1, D2, D3
13	<input type="checkbox"/>	<input type="checkbox"/>	2	MCP1700-3302E_T092	U2, U7
14	<input type="checkbox"/>	<input type="checkbox"/>	1	ATmega1284P-P	U1
15	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1	SN65HVD230	U3
16	<input type="checkbox"/>	<input type="checkbox"/>	1	MCP2515	U4
17	<input checked="" type="checkbox"/>	<input type="checkbox"/>	1	NAU7802SGI	U5
18	<input type="checkbox"/>	<input type="checkbox"/>	1	DS18B20	U6
19	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1	12MHz	Y1
20	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1	8MHz	Y2
21	<input checked="" type="checkbox"/>	<input checked="" type="checkbox"/>	1	Fuse/Polyfuse 500mA	F1
22	<input type="checkbox"/>	<input type="checkbox"/>	1	SolderJumper_2_Open	JP1
23	<input type="checkbox"/>	<input type="checkbox"/>	4	Conn_01x03_Male	J4, J5, J7, J8
24	<input type="checkbox"/>	<input type="checkbox"/>	1	AVR-ISP-6	J1
25	<input type="checkbox"/>	<input type="checkbox"/>	1	Conn_01x06	J2
26	<input type="checkbox"/>	<input type="checkbox"/>	1	Conn_02x05_Odd_Even	J3
27	<input type="checkbox"/>	<input type="checkbox"/>	1	Conn_01x08	J6



Plugins (2/2):
HTML BOM

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>



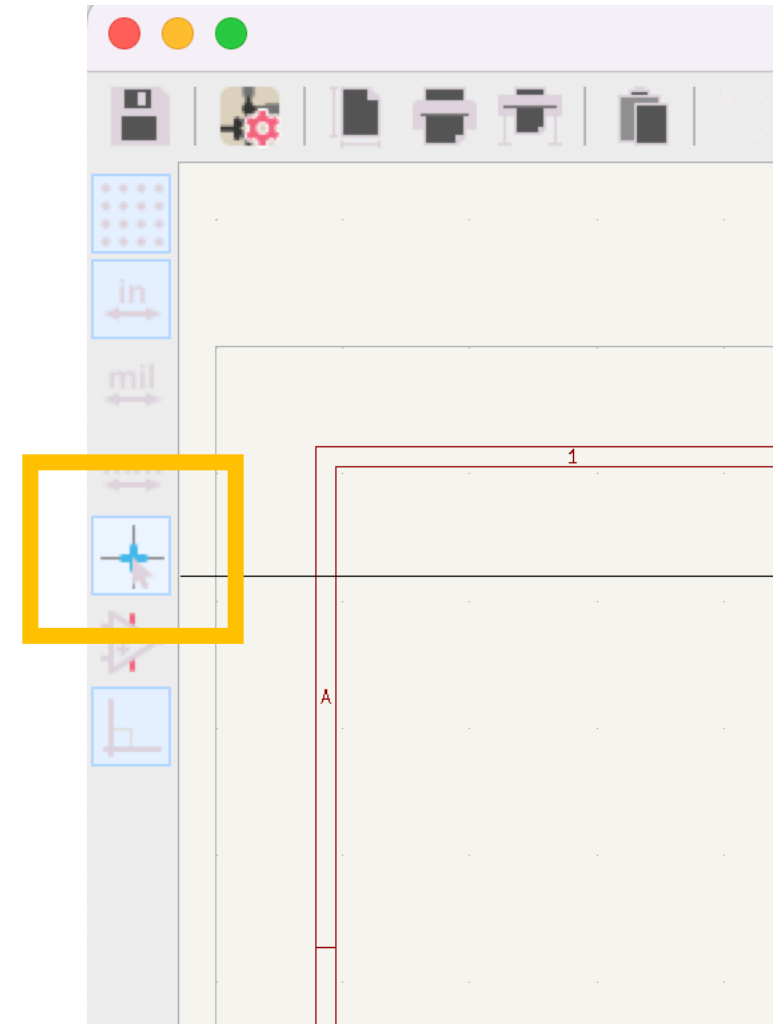
Since 2020: new procedure

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

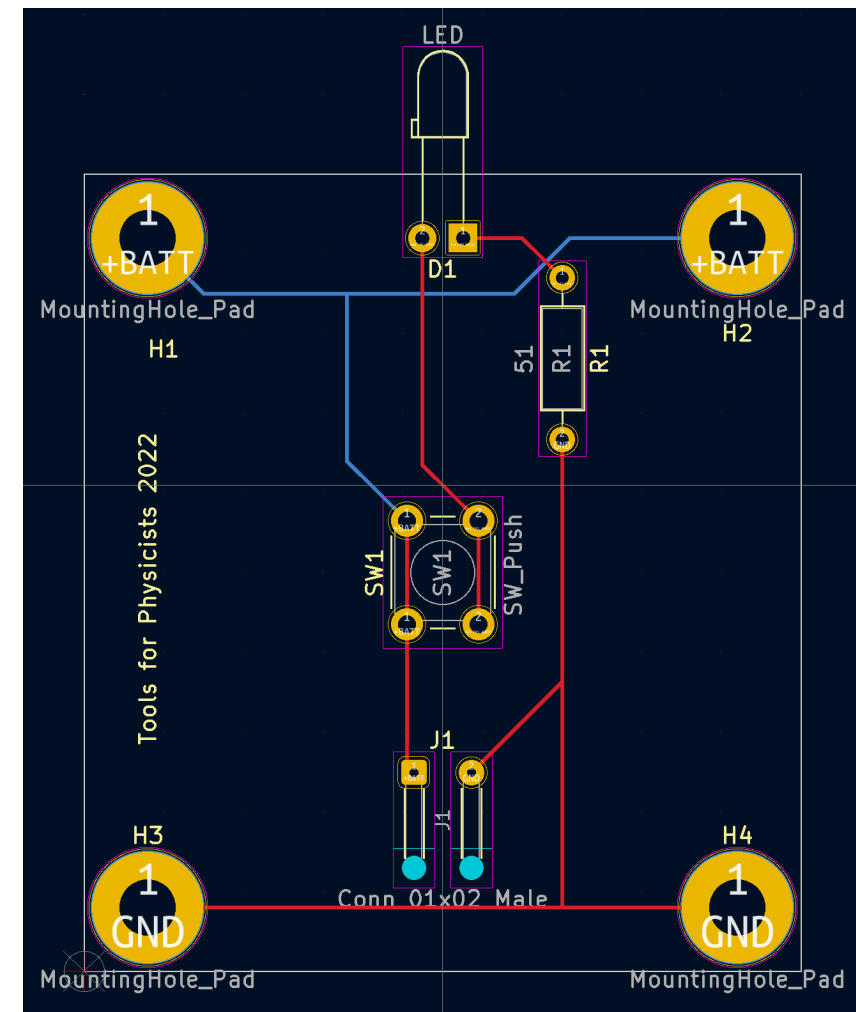
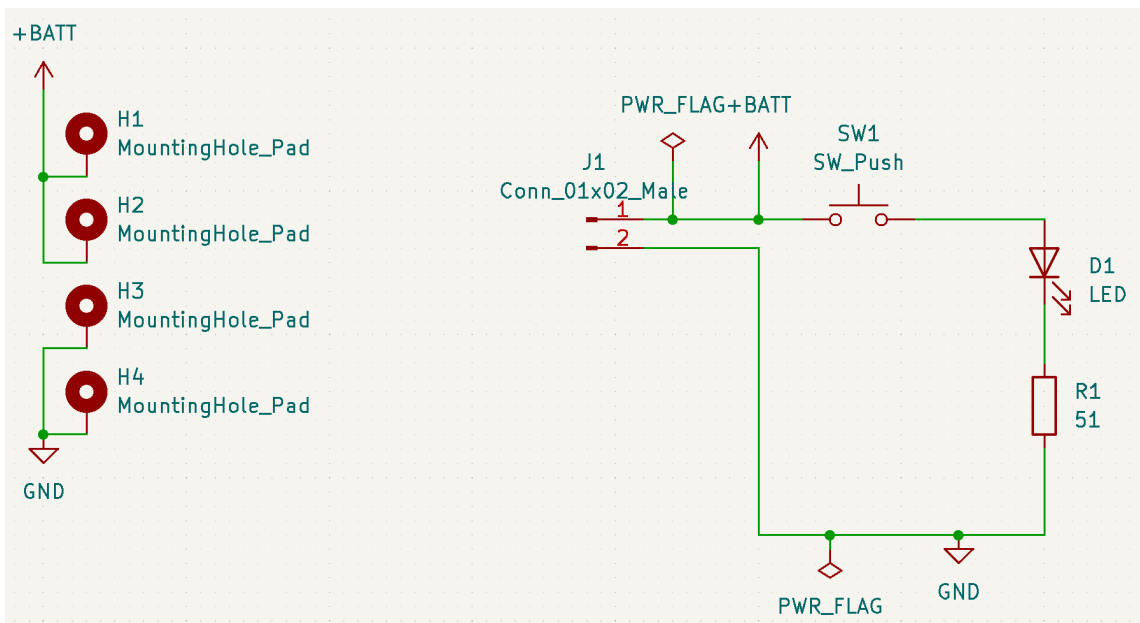
Hints for preferences

- 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/6.0/en/getting_started_in_kicad/getting_started_in_kicad.html#basic_concepts_and_workflow)
https://docs.kicad.org/6.0/en/getting_started_in_kicad/getting_started_in_kicad.html#basic_concepts_and_workflow

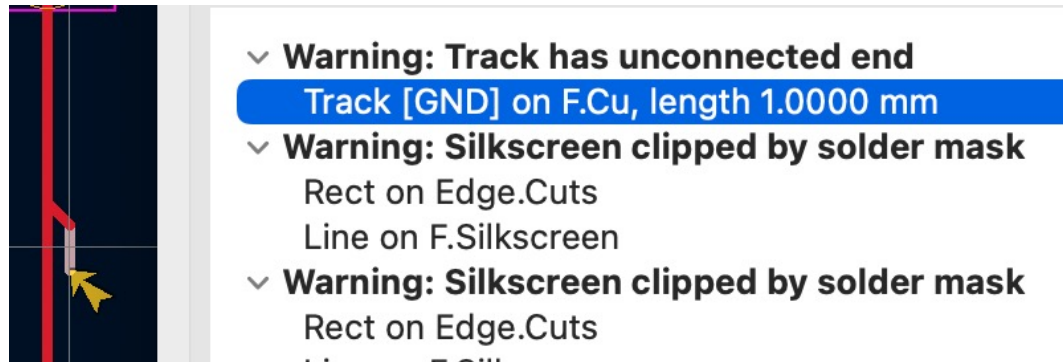


Symbol Fields table

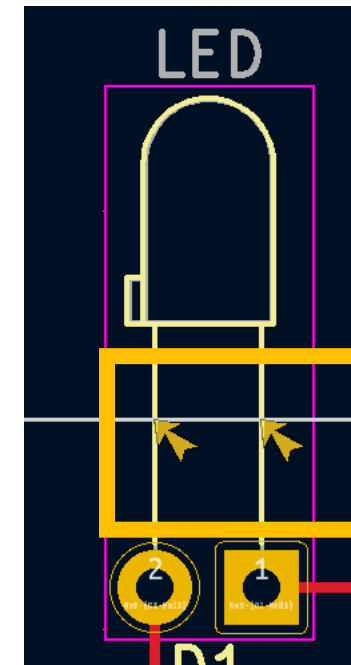
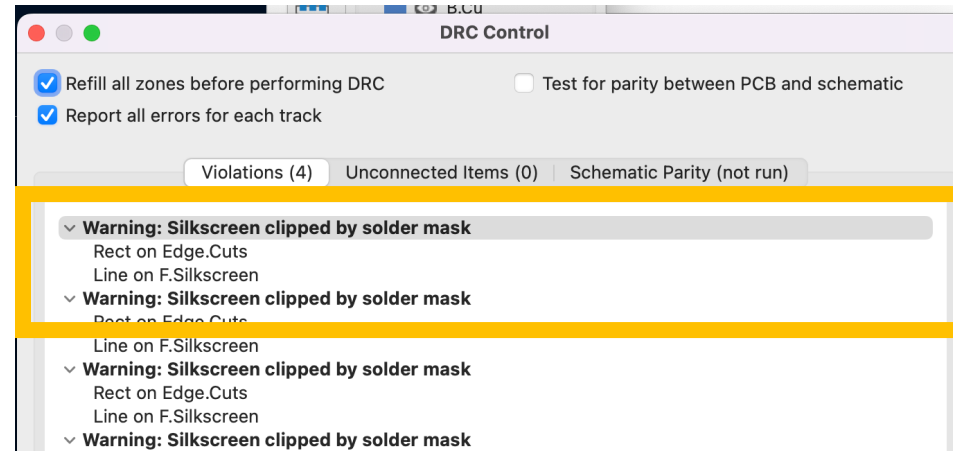
Reference	Value	Footprint	Datasheet	Qty
D1	LED	LED_THT:LED_D3.0mm_Horizontal_O6.35mm_Z2.0mm	~	1
> H1-H4	MountingHole_Pad	MountingHole:MountingHole_3.5mm_Pad	~	4
J1	Conn_01x02_Male	Connector_Wire:SolderWire-0.1sqmm_1x02_P3.6mm_D0.4mm_O	~	1
R1	51	Resistor_THT:R_Axial_DIN0207_L6.3mm_D2.5mm_P10.16mm_Ho	~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



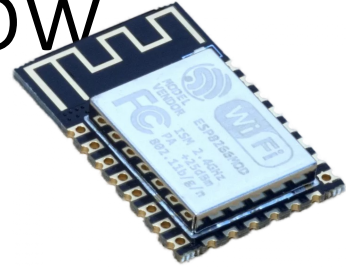
5) Custom Symbols and Footprints

- (Not for today's problem class)
- Detailed and up to date tutorial by the makes of KiCAD:
https://docs.kicad.org/6.0/en/getting_started_in_kicad/getting_started_in_kicad.html#creating_new_symbols

3 Step process:

1. Creating New Symbols
2. Creating 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.

