Introduction to PCB design

Claude Dahan

21st February 2025

École
polytechnique
fédérale
de Lausanne

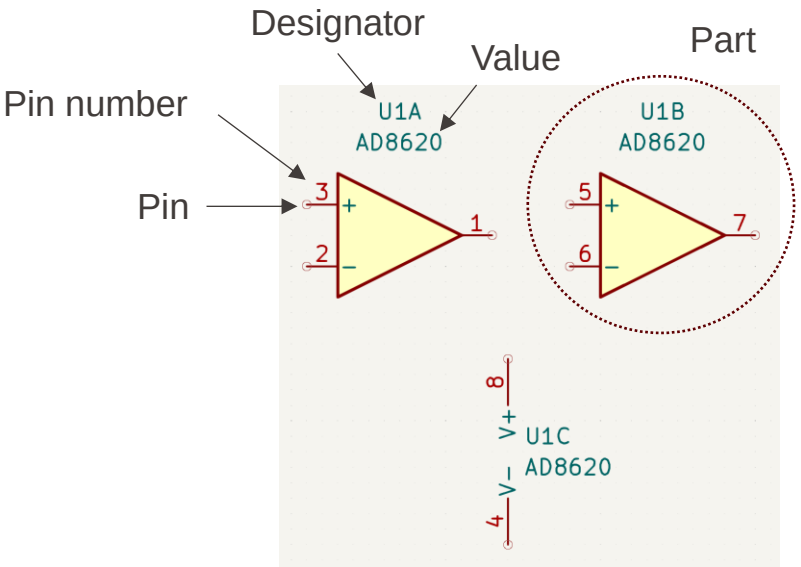
Software

- Electronic Design Automation (EDA), or ECAD
- Many choices with many capabilities:
  - Commonly used free software: KiCAD

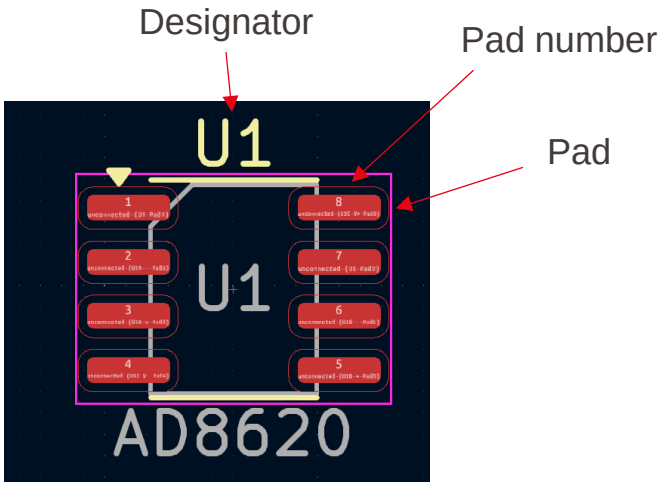
Of importance to us:

- Schematic and Layout design capabilities
- Checks:
  - Electrical Rules Check (ERC): Does the schematic make sense?
  - Design Rules Check (DRC): Can the PCB be fabricated?
  - Layout Vs Schematic check (LVS): Does the layout match the schematic? (sometimes part of the DRC)

Symbols & Footprints



Symbol(s)
But single component



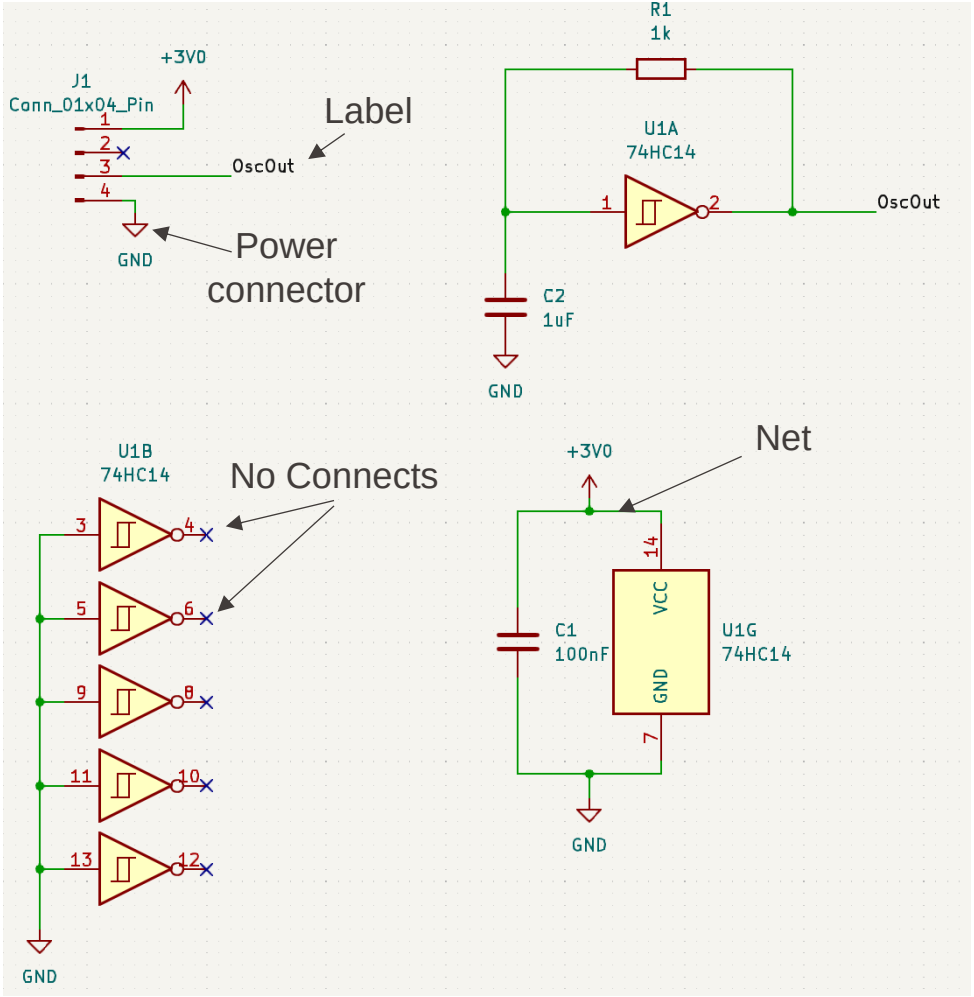
Footprint

Pin and pad numbers should match with the component used
(refer to datasheet)

Schematic example: Basic clock

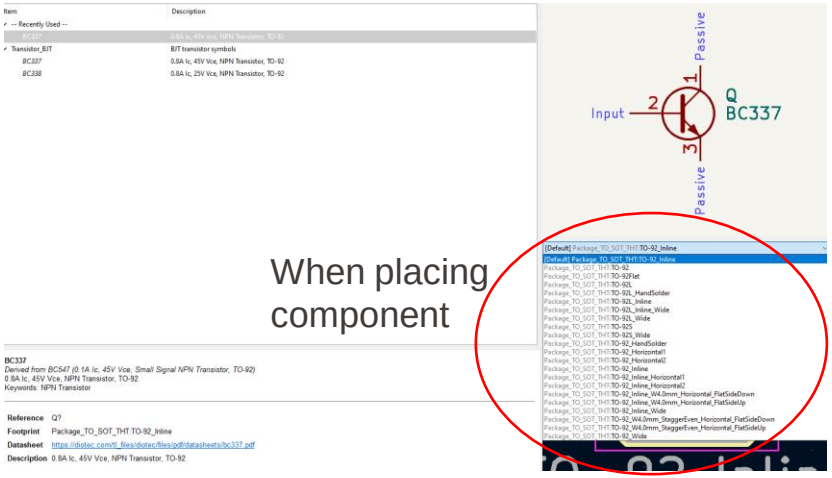
Tips & tricks:

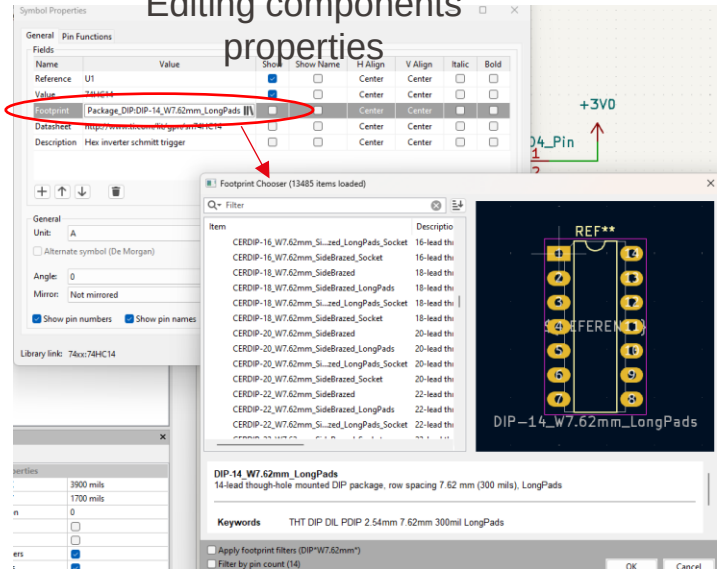
- Use net labels to link nets without wires
  (Possibility to have “global labels” to link between sheets)
- Use “No Connect” crosses when pins are unused
- Add decoupling capacitors to ICs
- Place unused parts on the schematic, tie inputs BUT NOT OUTPUTS (obviously)



Moving to PCB

- Make sure all footprints are already set in schematic
  - If footprint and/or symbol of your component does not exist, create it according to datasheet





Custom symbols and footprints

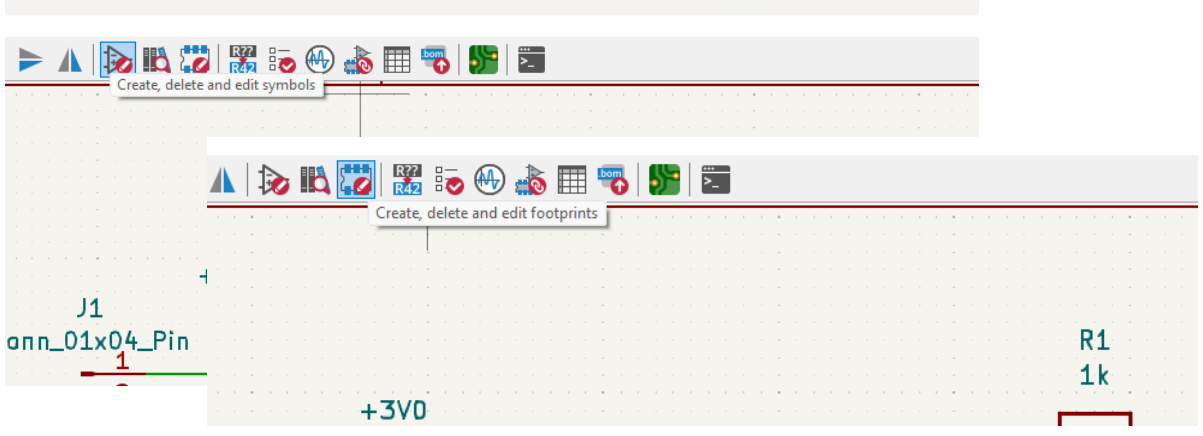
Default library should suffice in most cases, so only briefly shown during demo

Steps quite self explanatory:

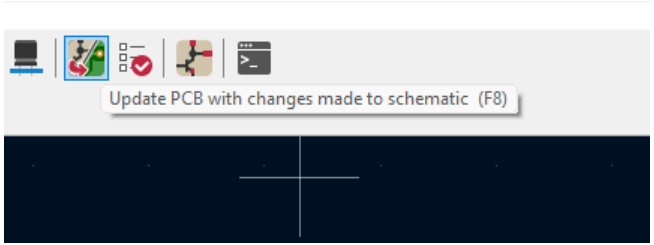
In create symbol window: File→New library
Select library
File→New component
Add pins, set designator, …

Many resources online if need be, or ask us ☺

Take care to match pins and pads number order!



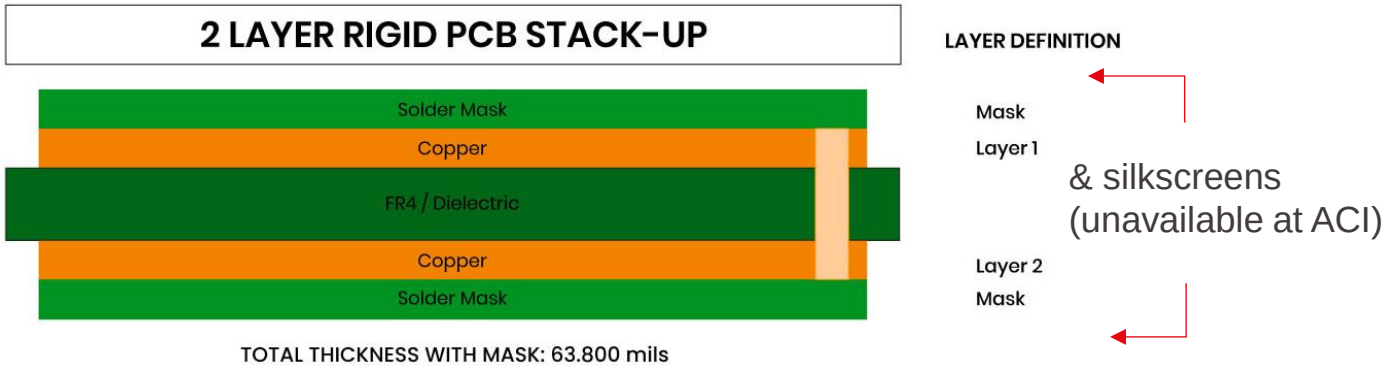
Importing in PCB

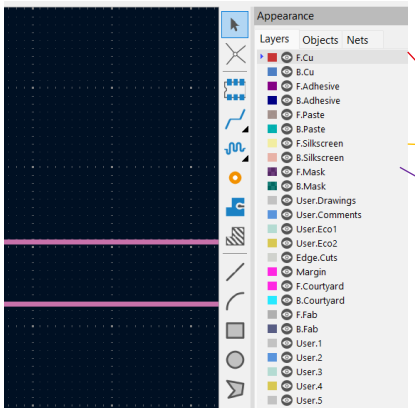


- Lists all the components/nets/… not matching between PCB and schematic
- All validated differences will be corrected on the PCB's side, adding components and connections alongside

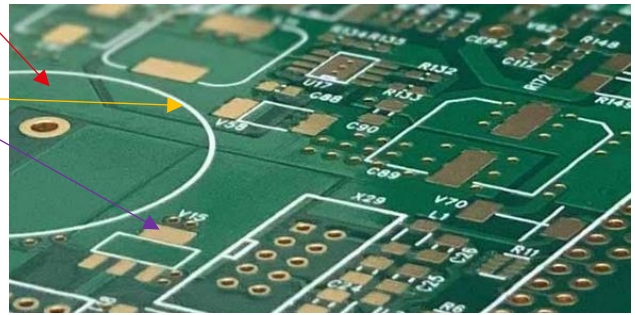
- Note: Importing from schematic only checks that the component exists. If the footprint was changed in a library, you need to "update footprints from libraries"

PCB stack-up





F. : Front side (more often called TOP)
B. : Back side (more often called BOTTOM)



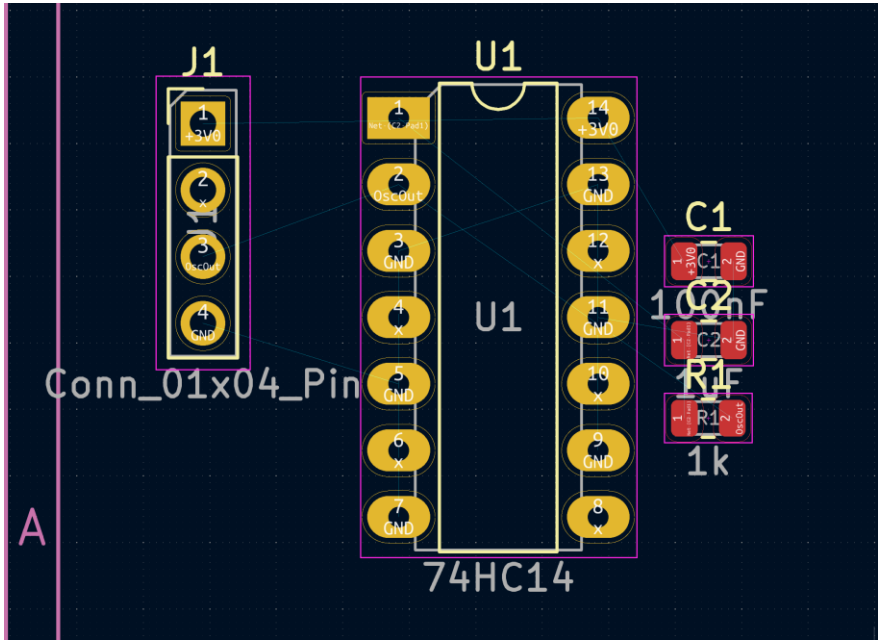
Cu: Where there is copper (tracks, pads, ...)

Paste: For solder paste, usually equal to “mask”
Silkscreen: For annotations
Mask: Where no solder mask should be placed

Edge cuts: Shape of the board
Fab: Fabrication schematic/annotations
...

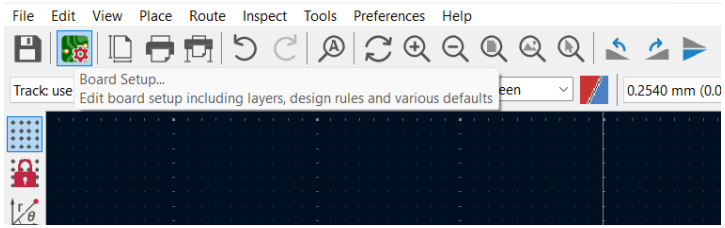
With components

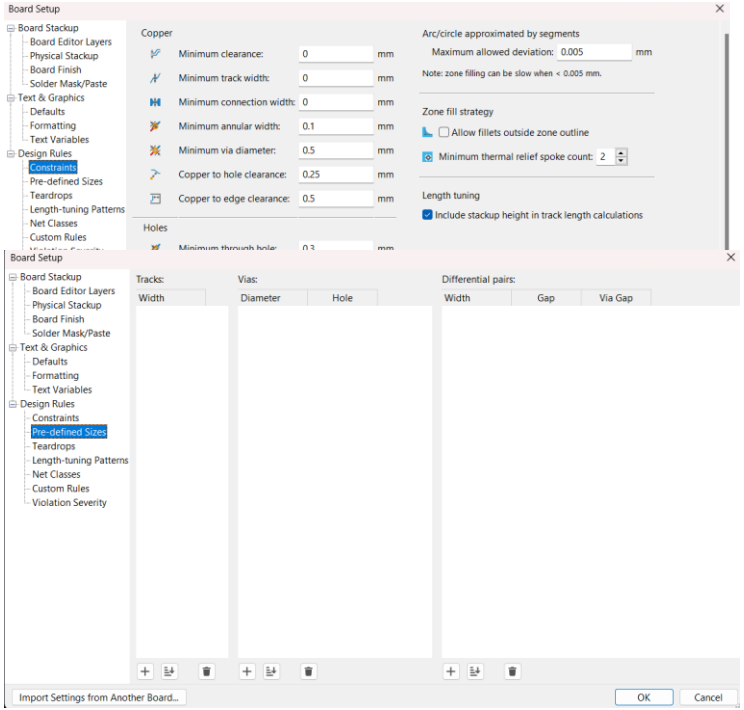
- Once everything imported from schematics, it's time to plan the routing
- Keep the final application in mind:
  - Decoupling caps close to ICs' power pins
  - Mechanical constraints (location of connectors, holes, ...)
- Ratsnest: small lines showing the connections remaining



Set board rules

- Prepare rules corresponding to fabrication constraints
- **Good to do early on**





Board rules

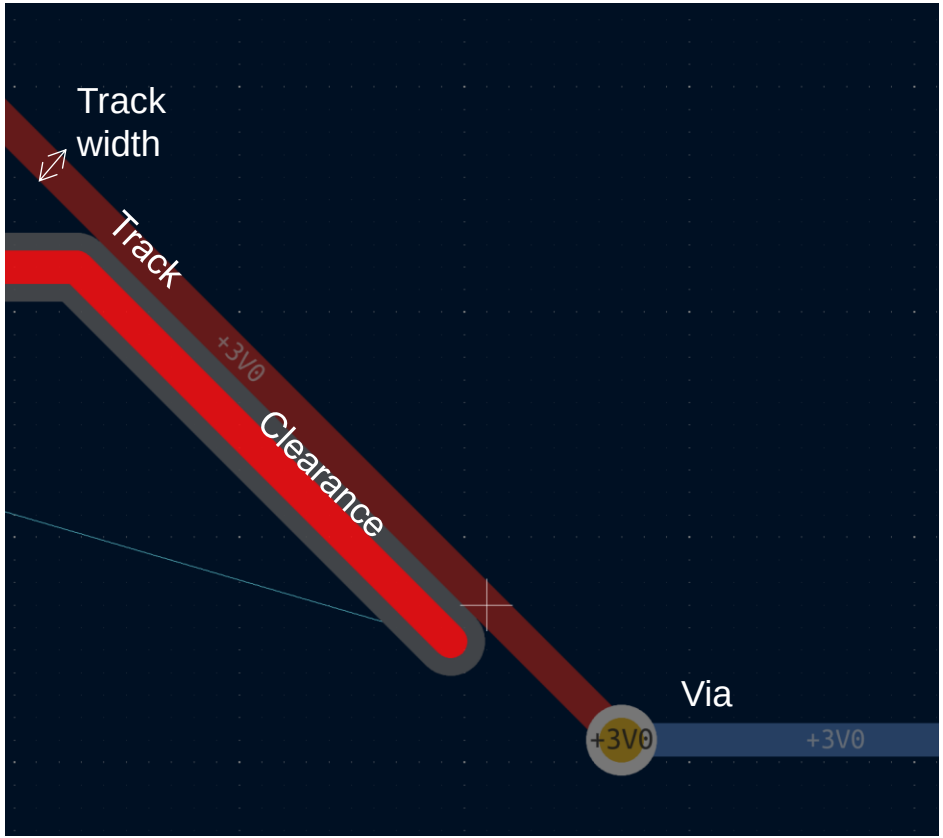
ACI gives set of rules.
(our PCB is made of FR4 0.8mm, 12um Cu)

Standard Dimensions (in mm)		
Description	**PCB size max.**	**Panel size max.**
Small	148 x 180	170 x 250
Medium	228 x 230	250 x 300
Large	228 x 310	250 x 380

Hole Diameter (in mm)		
Material Thickness	**Min. Hole Diameter**	**Max. Hole Diameter**
	(c)	(c)
0.8	0.3	6.2

Available drills:

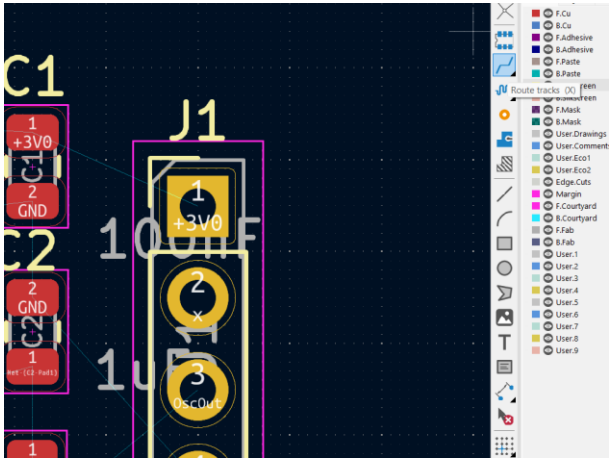
- From 0.25 up to 3.25 mm, by steps of 0.05
- From 3.30 up to 6.2 mm, by steps of 0.1
- Drill diameters above 6.2 mm must be placed in the Outline file, GERBER RS-274X format

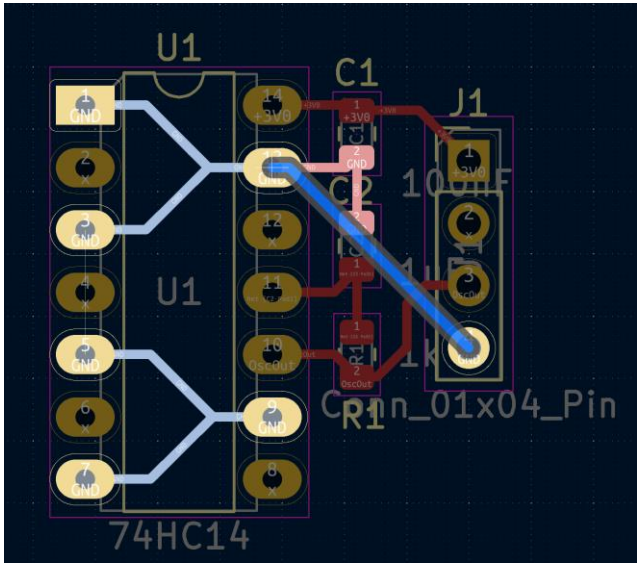


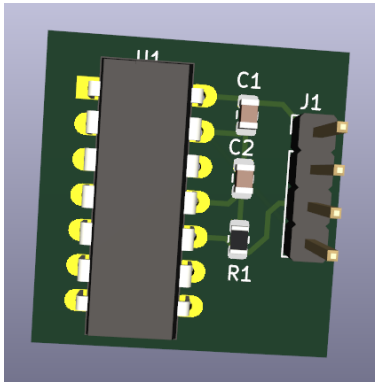
In brief, for our application:

- Width and clearance of minimum 250µm (10mil)
- Adapt width to current if high current application
- Via of 800µm, with hole of 500µm

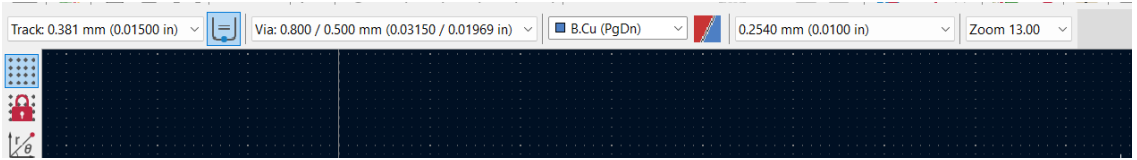
Routing







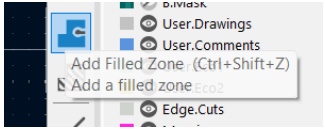
Use shortcuts:
Route (X), place vias (V), flip component to other layer (F), ...



Ground plane

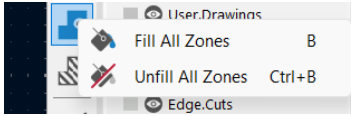
- Once the routing is finished, it is good practice to add a ground plane, to improve the shielding on the PCB, and reduce copper waste in fabrication

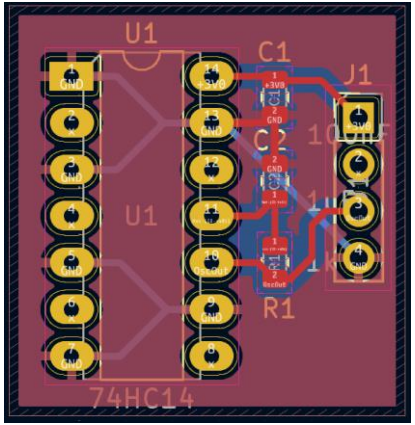
- Select a copper layer →



- Add a zone following the PCB's edges, on TOP & BOTTOM, tied to GND

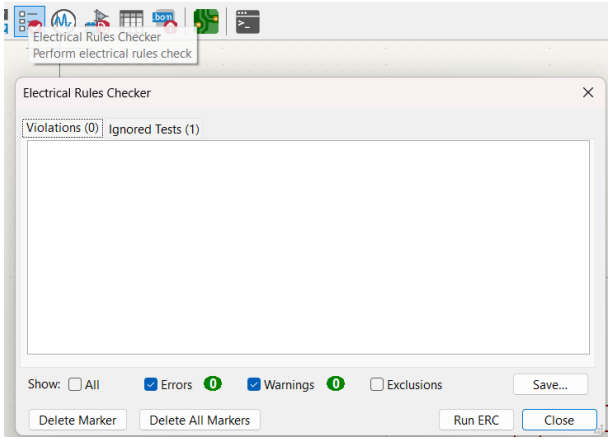
- You can "pour" it, with right click → or by running the DRC



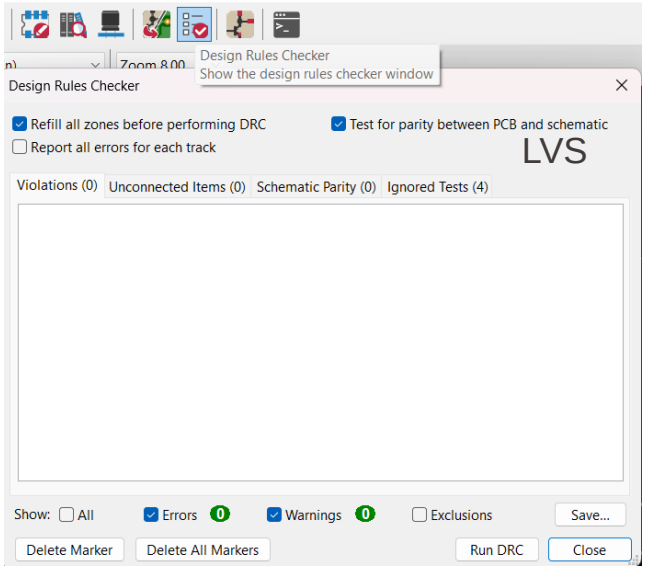


Checks

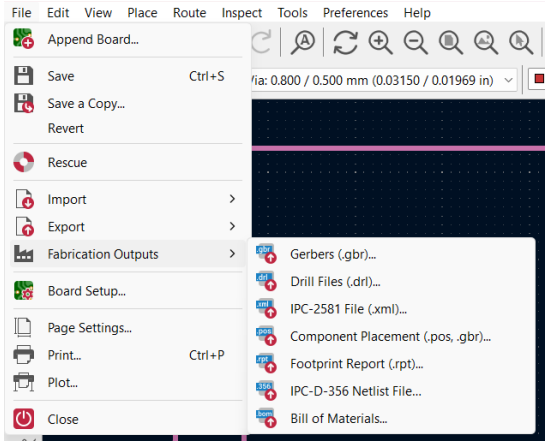
ERC

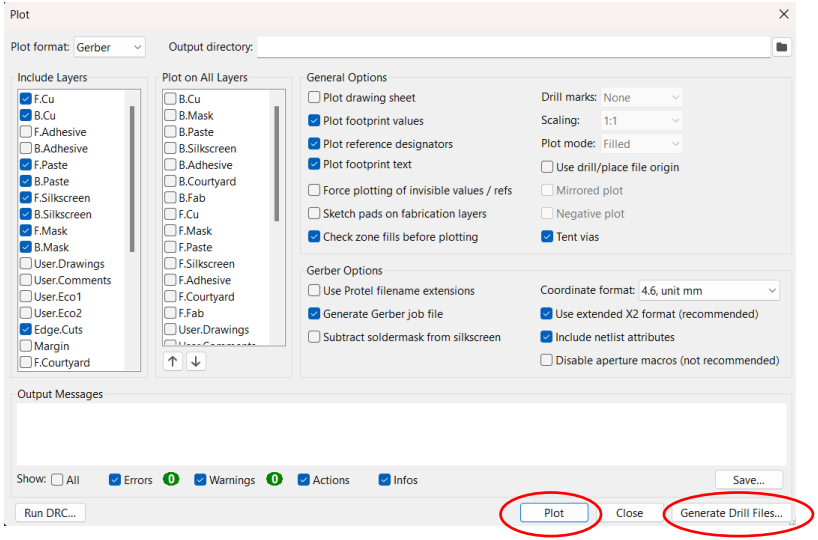


DRC



Gerber files





"Gerber files" are fabrication files containing each layer's pattern
To make the PCB, both Gerber and Drill files are needed

"Include layers" → One file per layer, as needed for fabrication
"Plot on all layers" → Useful to export a PDF showing the final PCB and verify by hand

To provide: **Gerbers** of **Copper**, **Paste** and **Mask** layers (no silkscreen needed)
Drill Files (Excellon, other parameters should not matter as defined in the file itself)
PDF to scale of PCB layout