UNIVERSITY OF OXFORD

Advanced Instrumentation Lectures: PCB Design with KiCad

Dan Weatherill

Oxford Physics Microstructure Detector Laboratory

Outline

UNIVERSITY OF OXFORD

Session 1 (Tuesday)

- **Recap on PCB basics (see Sneha's lecture)**
- **Intro to KiCad**
- **Comparison of KiCad to other programs**
- **Quirks / drawbacks to be aware of in KiCad**
- **KiCad workflow options – symbols, footprints etc**
- **Component suppliers & BOM considerations**
  - **RS, Farnell, Mouser, Digikey**
  - **Octopart**
- **PCB Manufacturing Houses**
  - **Newbury/PCBTrain (UK)**
  - **The "boutique" shops (UK)**
  - **China: example Seeedstudio Fusion**
- **KiCad install & Schematic Editor demo**

Session 2 (Friday)

- **KiCad Schematic Editor demo (continued, with SPICE)**
- **KiCad PCB Editor demo (routing, netclasses)**
- **Exporting for Manufacture (gerbers, BOM)**
- **Exporting for mechanical CAD**

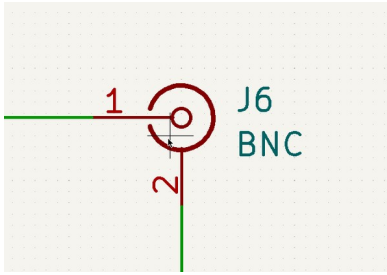
PCB Basics recap

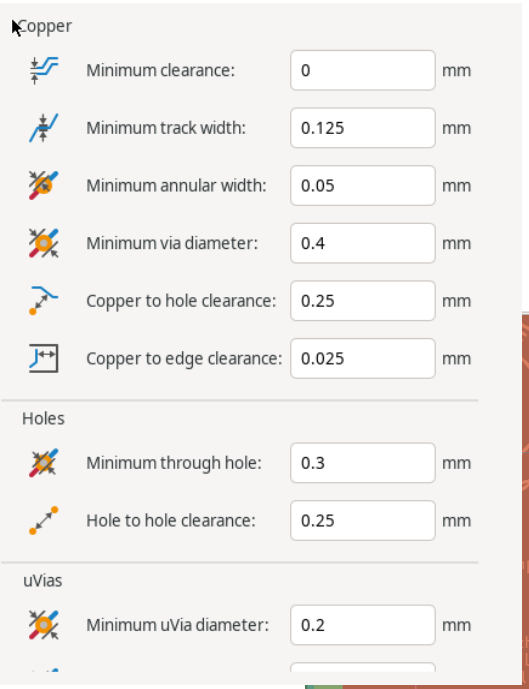
UNIVERSITY OF OXFORD

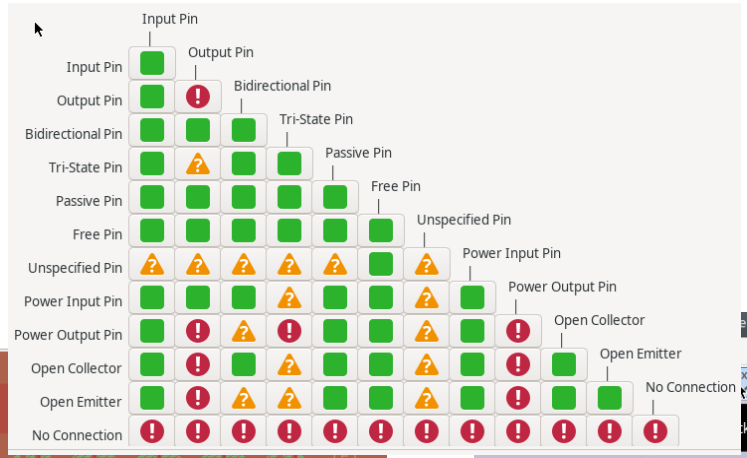
Talk by Sneha Naik earlier in this series:

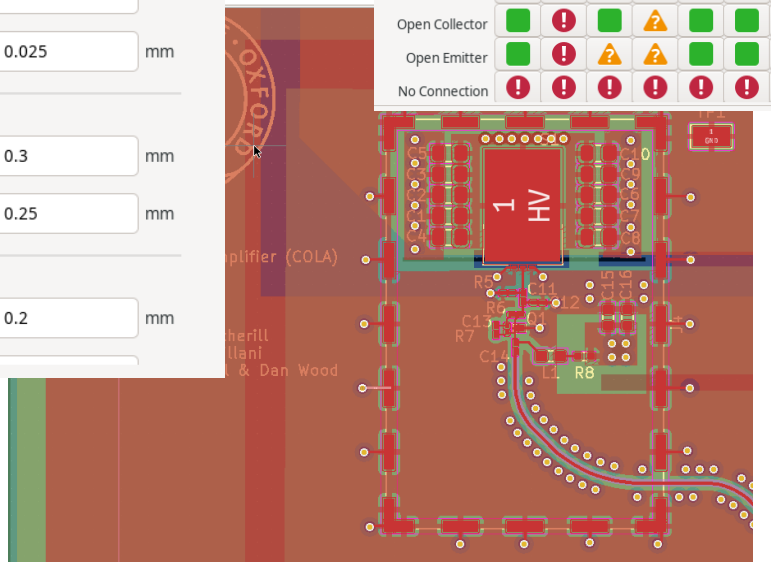
https://indico.cern.ch/event/1277888/contributions/5367707/

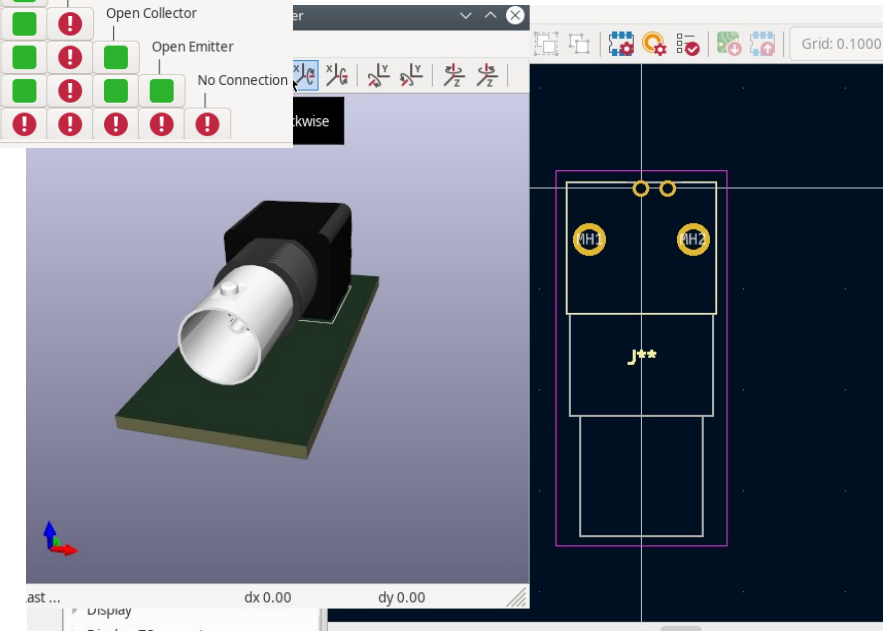
Make sure you know the basics of the following words (ask if any are unclear):
Schematic, Board artwork, Symbol, Footprint, Layer, ERC, DRC, BOM











Intro to KiCad

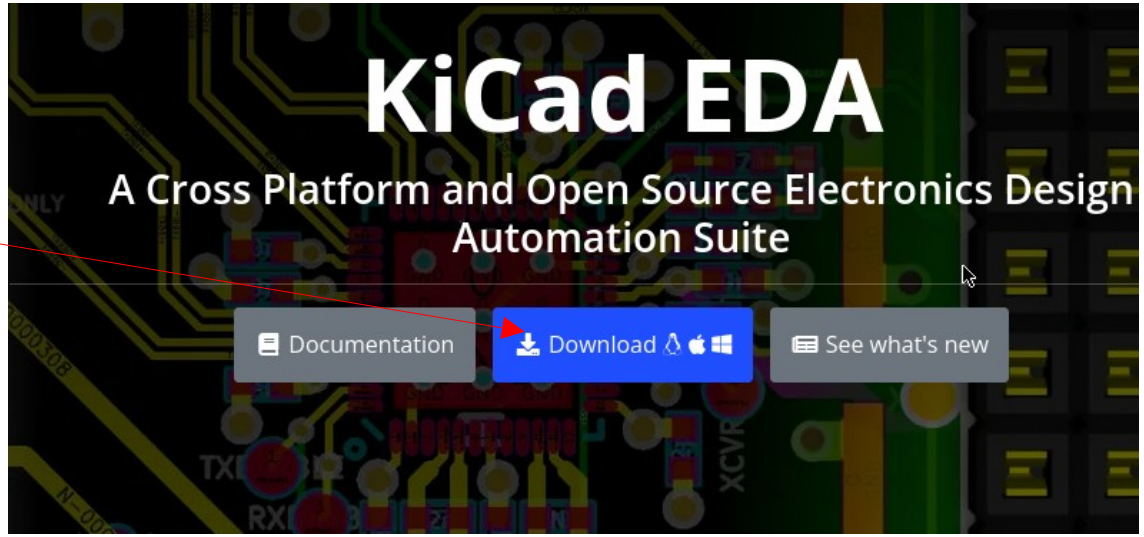
UNIVERSITY OF OXFORD

Get kiCAD here (we're using version 7)

https://www.kicad.org/

Good place to get help on KiCAD usage:

https://forum.kicad.info/



KiCad (pronounced "key-CAD" by original authors) began development in 1992 (!!) by Jean-Pierre Charras. Since ~2015 managed by Wayne Stambaugh + a team of developers (including at CERN). It is open source and you can contribute fixes & code.

Probably the best known, supported and widely used free, open source PCB design software.

Please try and download and install now if you Haven't already

CERN contribution:
https://ohwr.org/project/cern-kicad/wikis/home

Other Programs vs KiCad

UNIVERSITY OF OXFORD

AUTODESK® EAGLE

Very popular, free version available. Similar in features to KiCad (more advanced in some ways). Widely supported

But: every new version since Autodesk takeover gets more “locked down”. Free version limited in layer count & board size. Kicad can (quite well) import Eagle projects

See also: diptrace, designspark

Much more advanced “professional level” features that Kicad lacks (controlled impedance, stackup management, integrated parts database etc etc). Very good interface but with some learning curve.

Altium Designer

Recommend starting with a simpler tool (like KiCad) to design some simple boards and perhaps pick up something like Altium later. Kicad can import Altium projects though it is still quite an experimental feature

See also: OrCAD / Cadence Allegro, Mentor Graphics (PADS)

Interoperability

UNIVERSITY OF OXFORD

From mechanical CAD, you may be familiar with STEP / IGES, which allow (reasonable) interchange of designs between programs & for manufacturers. You need to be (vaguely) aware of some of this as it may affect exchanging designs with colleagues & fabrication houses.

For PCB design, the story is not quite as good. There is no good “schematic + PCB” export but there are some options:

- **Gerber “RS-274X/X2” –** the standard format that you will send to most board houses. Exports PCB artwork so it can be manufactured, but doesn’t really know anything about the circuit connectivity (no schematic information).
- **ODB++** - many “pro” level CAD programs support this and can exchange designs between them. **KiCad does not (yet) support it**, it is planned for version 7
- **IPC-2511b “GenCAD/GenCAM” –** similar format to ODB++ but less widely adopted, though it is an open standard. **KiCad can export an old version of GenCAD**, but it is not very widely used. **If you use GenCAD export in KiCad you should check the output it generates carefully with an external tool! (for example https://www.circuitcam.com/download)**
- **IPC-2581 –** “next generation” export format, which (may) eventually replace Gerber. Planned for KiCad to support it, **but it doesn’t yet**! **This is a problem for some (very few!) manufacturers, who now demand IPC-2581 rather than Gerber (see later)**
- **IPC-D-356A –** this is a netlist (rather than PCB artwork) format. Some board houses that do electrical test want it, KiCad **can export this**
- **Gerber “X3” –** the latest Gerber spec includes information similar to IPC-2581. Kicad **fully supports Gerber X3** . Sadly, most other programs do not, as it has not gained as much popularity as IPC-2581.
- Some board fabrication houses (e.g. in China) now accept KiCad project files directly! However, most of them don’t yet accept version 6.0/7.0 (only version 5.0), so ideally **stear clear of this!**

Things to beware of in KiCad

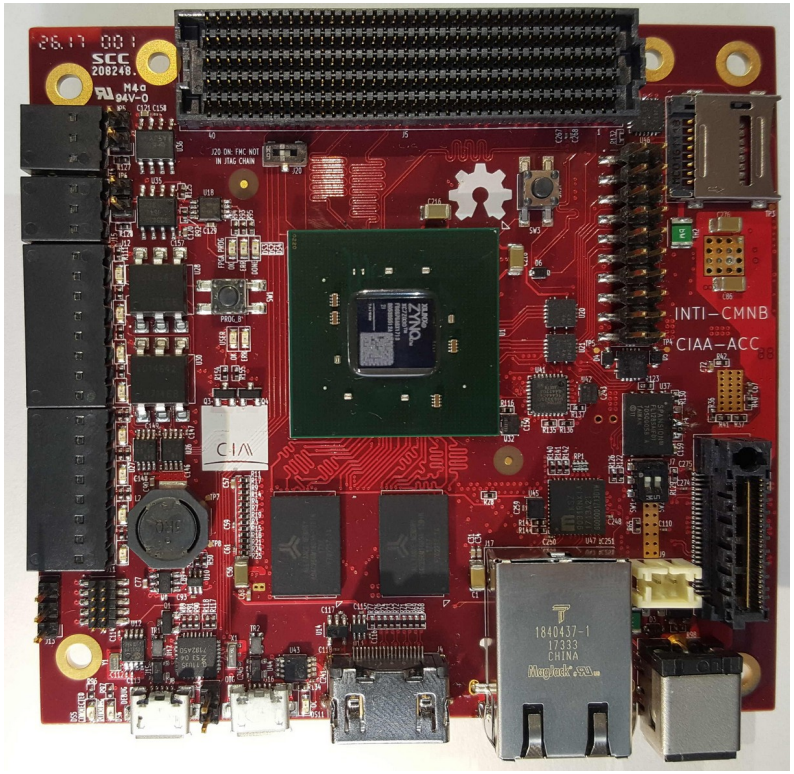
- **Not much physical layer stackup Awareness**
  Actually, KiCad 6 does have layer stackup now! And it can be included in basic constraints. BUT, not very good support yet for automatic controlled impedance traces (they can be done manually with some effort though!)
- **Have to choose a library workflow**
  Disagreements in community about whether to use “atomic” or “default” type parts (see later). Have to be careful with either choice (can mix & match to some extent)
- **Lack of IPC-2581 export**
  Some board houses want this now. It isn’t there yet in KiCad
- **Backward/Forward Compatibility Issues**
  Old versions of KiCad cannot open newer file versions. But new versions can import and migrate old projects very well
- **Some functionality needs plug-ins**
  Many of the really useful and advanced stuff is in community plugins. Over time more has moved into core (e.g. curved traces are now built in, but things like RF via stitching are still done through a plugin)
- **Included “standard” libraries are often not very good**
  The built in standard part library only includes a weird and narrow selection and KiCAd is not as good as Eagle or Altium when it comes to getting new parts. They have to be manually downloaded, rather than a nice “Wizard” dialog that you get in Altium
- **Grid system in Schematic entry is errr….**
  Well actually it’s much better in 6.x/7.x than 5.x, but still annoying at times

...And the Good News!

UNIVERSITY OF OXFORD

- **Huge community of users** – lots of people on the forums, IRC, around and about use this software
- **Open Source, no limits on "free" version** – no limits on board size, layers etc etc
- **Usable on "real" designs and in industry** – though it is a bit more limited than e.g. Altium, make no mistake KiCad is a professional grade tool

- **Text based file formats –** compatible with git version control! (And visual diff tools available, so merging is possible!)
- **Scriptable** – you can write plugins and processing tools in the python language to do custom features in KiCad.
- **pretty good (manual) routing experience**- advanced push'n'shove and other modern routing techniques are very well implemented and very usable. Better IMO than Eagle or Diptrace
- **No built-in autorouter** – autorouters almost ALWAYS get everything wrong anyway. An external autorouter (FreeRouting) is available if you really want it



(right) – CIAA-ACC 12 layer board with Xilinx 7000 Series SOC and kintex-7 FPGA. Running at GHz speed, A "very serious" design" done in KiCad!

KiCad Part workflow options

UNIVERSITY OF OXFORD

Kicad has no “built in” concept of a **part** (i.e. **footprint + symbol**). You can associate any schematic symbol with any footprint! This can be useful for quick designs and parts available in alternate footprints but **it is easy to make very bad errors which ERC will not save you from!**

Many serious people refuse to use KiCad for this exact reason (same applies to Eagle and, especially, DipTrace). Personally I don’t mind it but have fallen prey to a pinout error a couple of times over the years!

Luckily, if you want to use **atomic parts**, KiCad can support this! Tools such as componentsearchengine (see later) let you download parts in kind of atomic format

Atomic parts

Every different part (including different footprints of the same part), has a separate symbol & footprint in your KiCad library. Each symbol is associated to one (and ONLY one) footprint.

Pro: it’s very hard to make a very silly pinout mismatch error

Con: maintaining this parts library is a HUGE amount of work

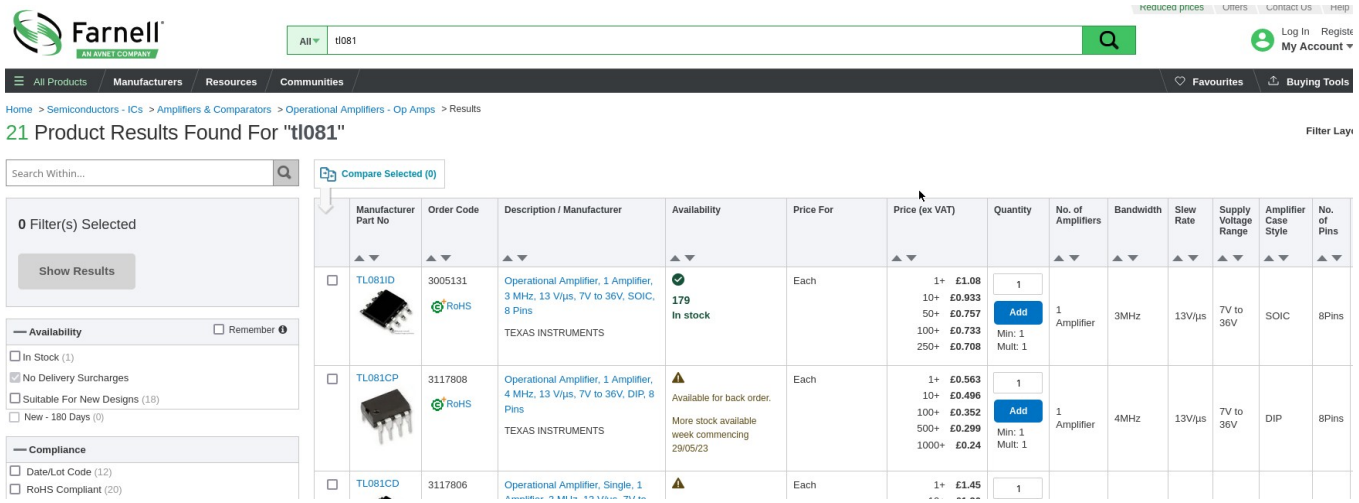
KiCad default

Each symbol in the schematic must be manually associated with a footprint when you design the board (but usually they have a default)

Pro: much quicker to get up and running, easier and more flexible to alter designs later
Con: you MUST DOUBLE CHECK FOOTPRINT ASSOCIATION VERY VERY CAREFULLY

Buying Components - UK

Sorting out the BOM is (usually) actually the most tedious part of a PCB Design project!
My personal recommendation – start with Farnell! (https://uk.farnell.com/)
Good search interface, wide selection, reasonable price.

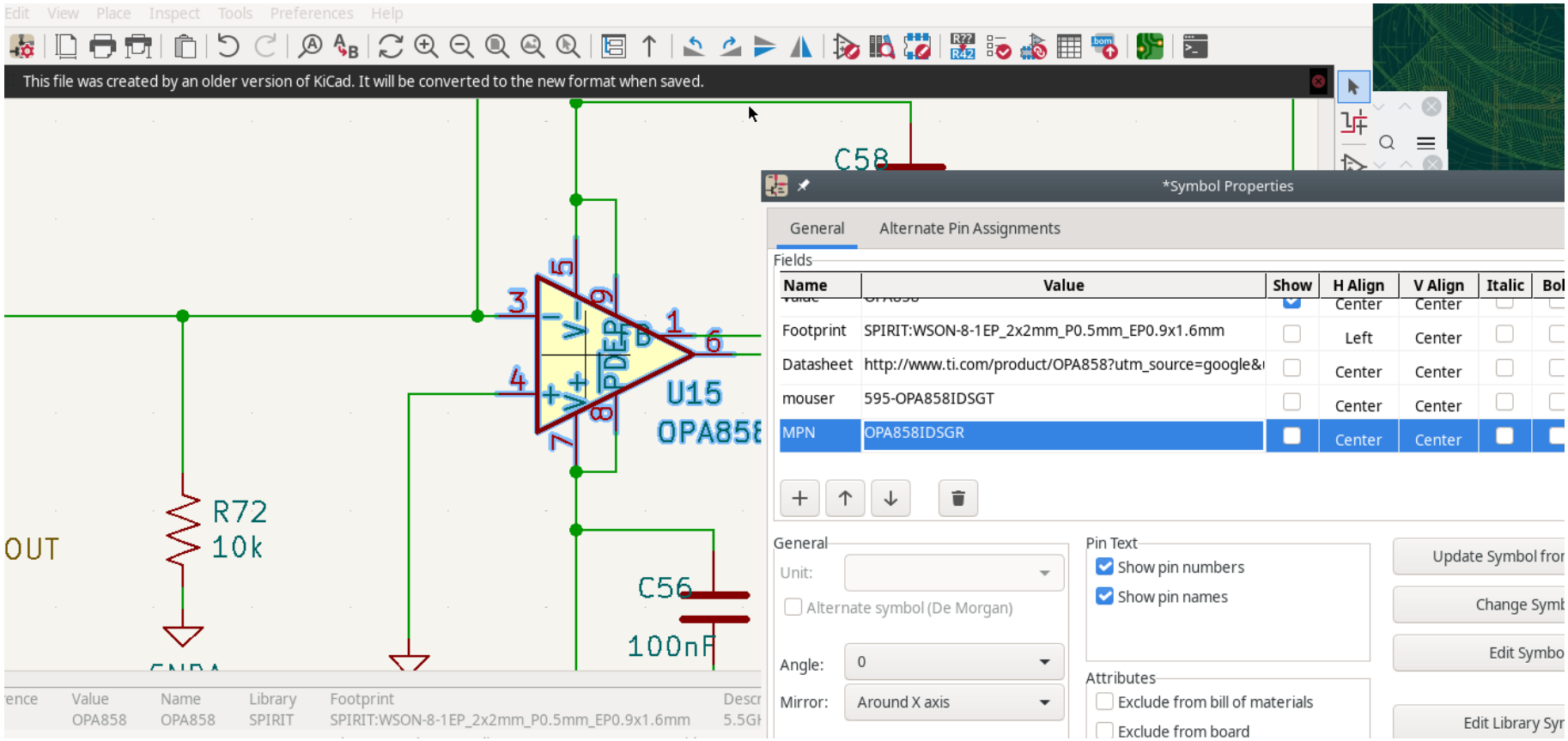


If a part is not on Farnell, it is likely on Mouser (www.mouser.co.uk) or digikey (www.digikey.co.uk). In some circumstances, it might be on RS (https://uk.rs-online.com/web/)

STRONG RECOMMENDATION: include both MPN (“manufacturer part number”) and the order code for a particular supplier in your schematic. Then, stock issues can be more easily sorted out via OctoPart or by the board assembly house

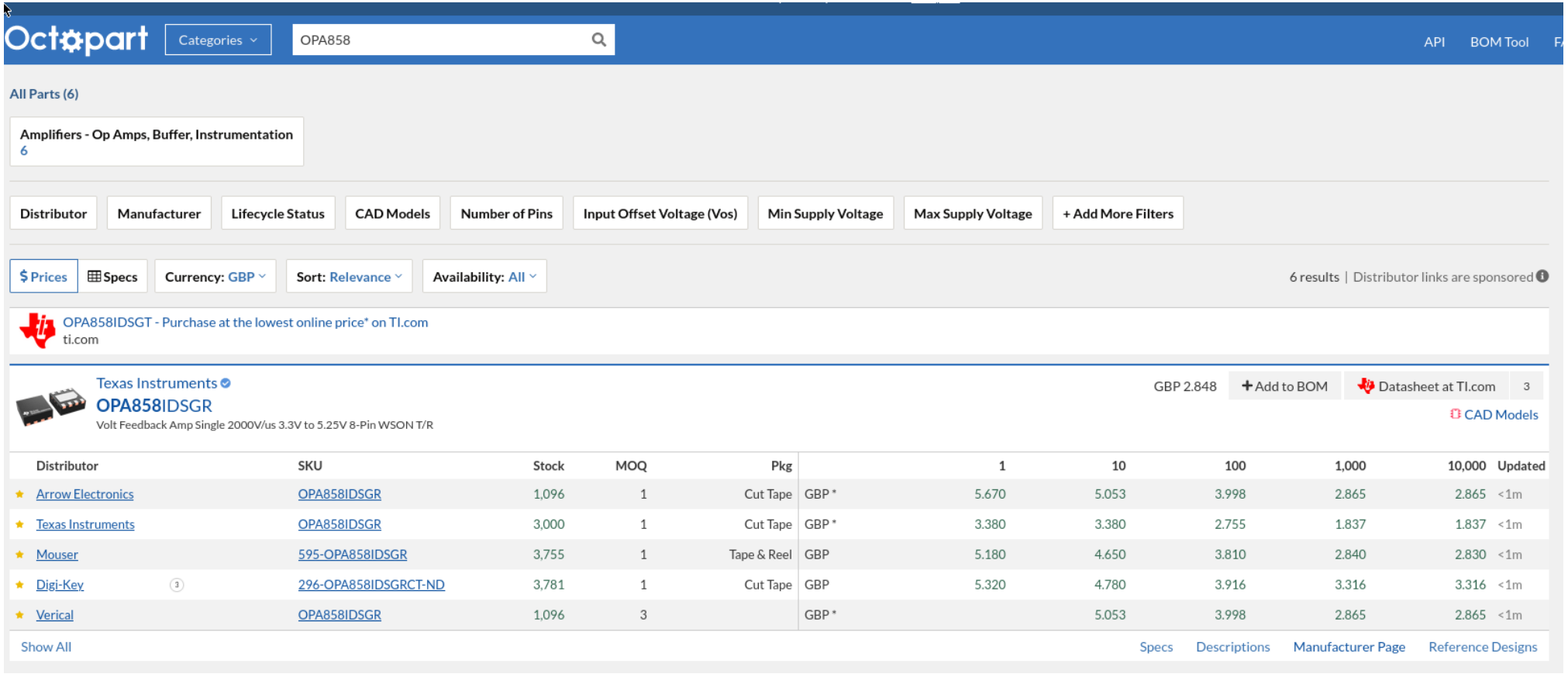
Octopart

Always put a manufacturer part number (MPN) in your schematic data!
You can use the tool www.octopart.com to find this part on many different suppliers, and sometimes to obtain CAD data downloads.



Octopart

UNIVERSITY OF OXFORD



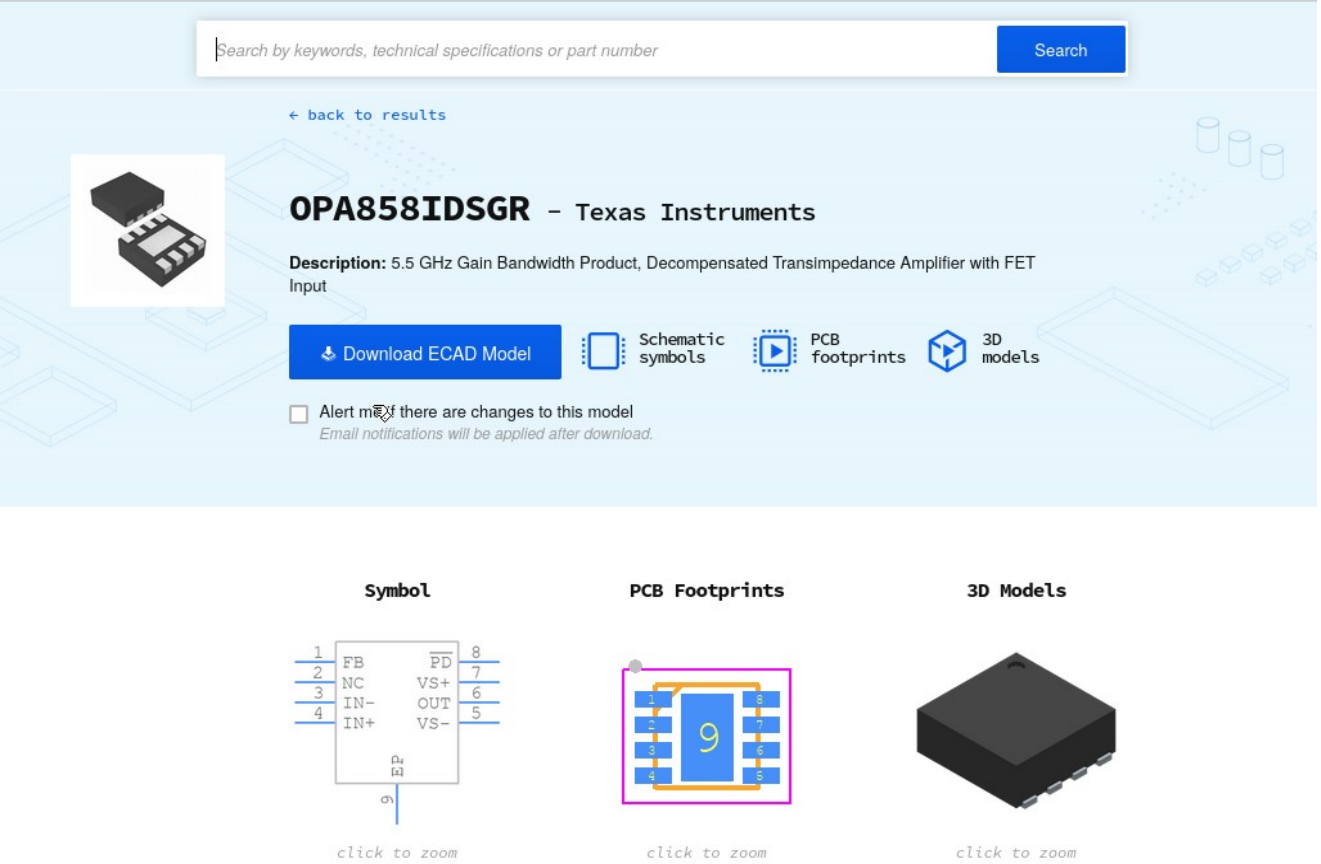
There are various BOM tools and plugins available for KiCAD that can find supplier part numbers and price a BOM from the MPNs in your schematic.

Component Search Engine

UNIVERSITY OF OXFORD

https://componentsearchengine.com

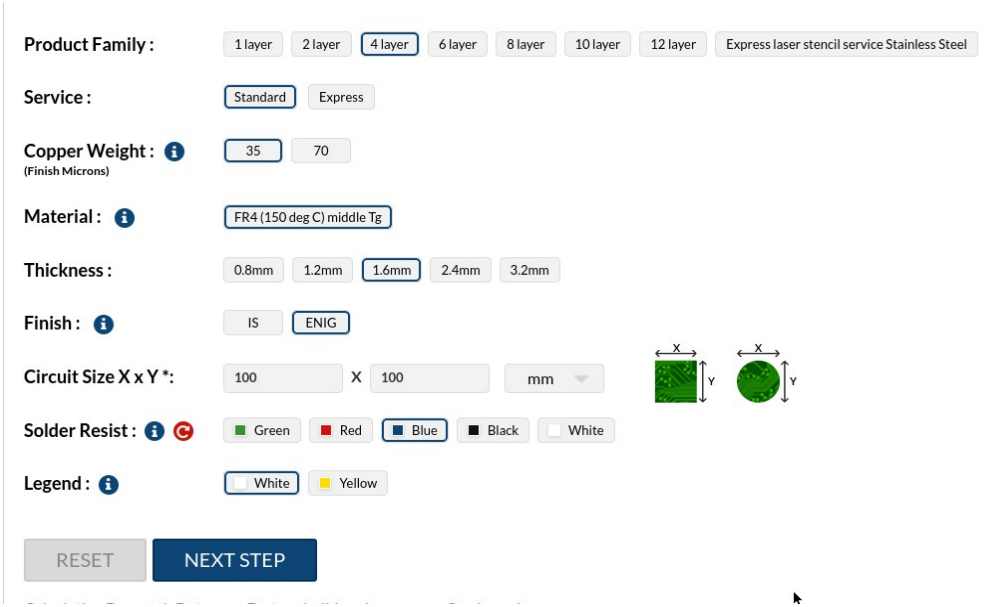
Very useful tool to obtain (mostly basic but usable) Footprints, symbols and 3D models of parts that are not in KiCad's built in libraries (**which is very often!)**

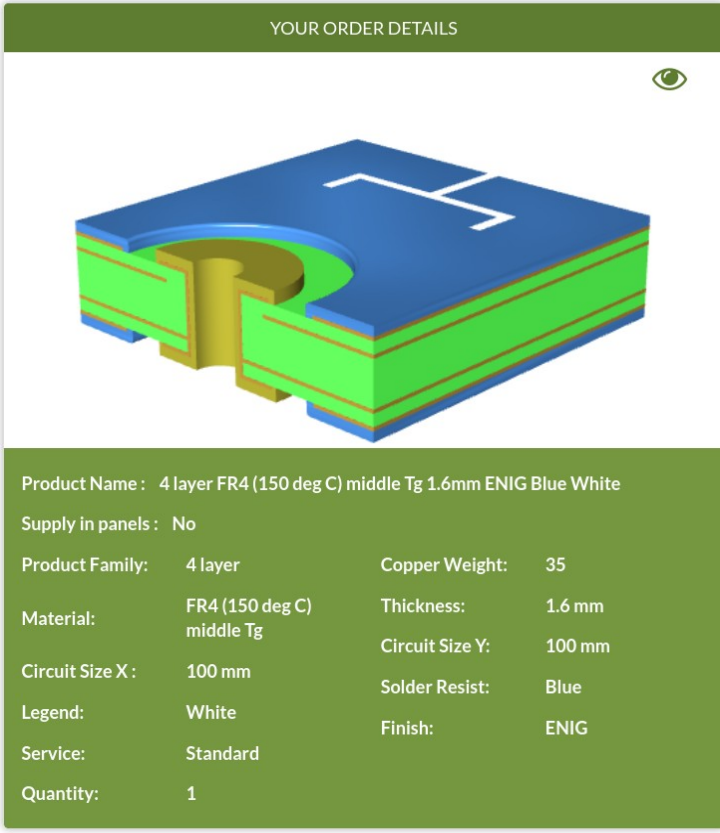


Manufacturing a PCB in the UK

UNIVERSITY OF OXFORD

Reasonably priced, fast turnaround available, online quotes that are pretty good: https://www.pcbtrain.co.uk (a.k.a Newbury Electronics). Can do fabrication and assembly, upload your design and BOM details and get a quote in your inbox in a couple of minutes





Price in GBP, exclusive of carriage

W/Days	1 Qty	2 Qty	3 Qty	4 Qty
4	£ 425.55	£ 476.88	£ 528.21	£ 579.52
5	£ 312.71	£ 357.28	£ 401.85	£ 446.44
7	£ 278.44	£ 317.62	£ 356.79	£ 395.96
10	£ 227.04	£ 258.10	£ 289.17	£ 320.24
15	£ 184.21	£ 208.52	£ 232.83	£ 257.16

Manufacturing a PCB in the UK

A range of more expensive & boutique manufacturers who can do more custom things and guarantee to verified IPC quality levels, including electrical test etc.

Careful, these get very expensive very fast.

Word of warning: **express circuits**, whilst a very good manufacturer (in my experience), now REQUIRES your data in non-gerber format, and as such right now it is **hard, though not impossible** to directly send them designs from KiCad.

This will improve within a few months hopefully.

Recommendation for “boutique” manufacturer: Cambridge Circuit Company.

Cost typically ~2-3x as much as Newbury

Manufacturing from China

UNIVERSITY OF OXFORD

Chinese “prototype” level manufacturers, including e.g. JLPCB and OSHPark should **not be discounted.** The manufacturing quality is typically very high, and cost much lower than UK manufacturers. Shipping is generally not that bad either. **However**, be prepared for them to manufacture **exactly** what you send them, mistakes and all!

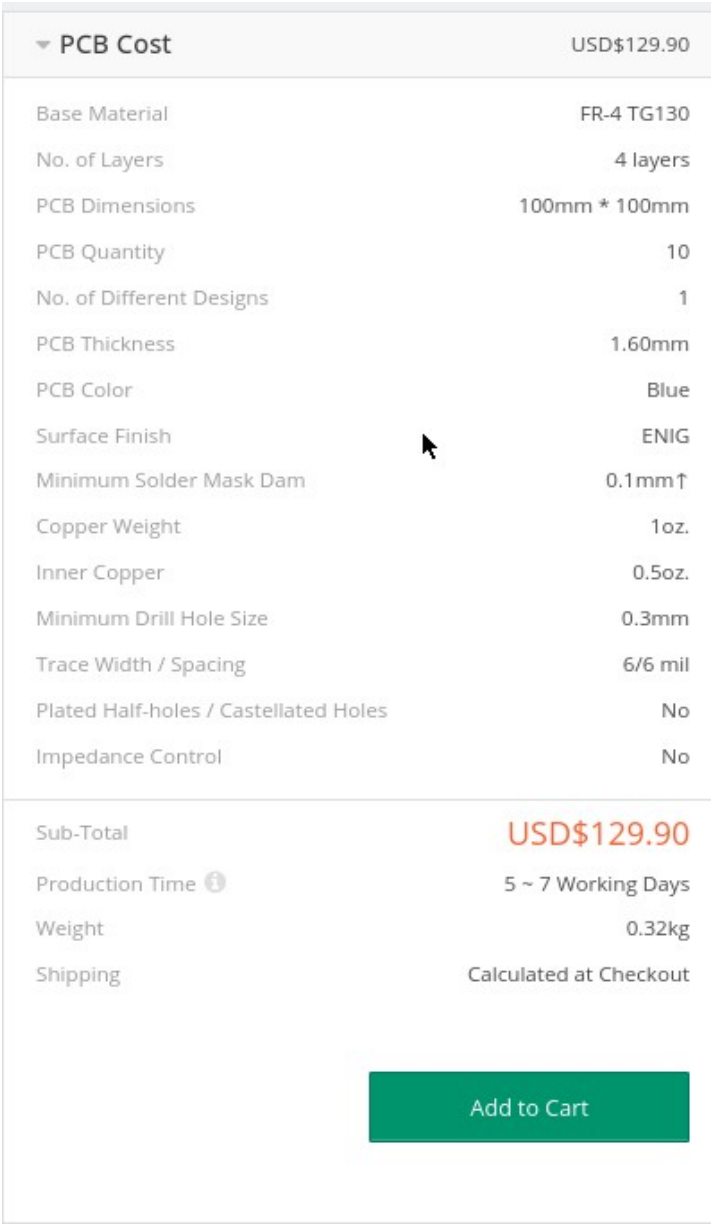
You will not get an email saying “are you really sure you want this”?

Example: my personal favourite based on experience, SeeedStudio fusion (http://www.seeedstudio.com/pcb-assembly.html)

(Right: ~same spec as Newbury/PCBTrain)

Newbury: 10 off in 10 w/days £506.60

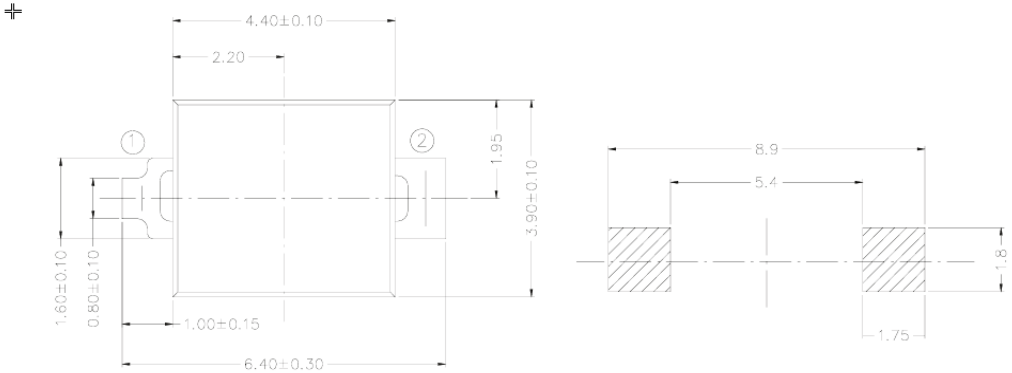
Seeedstudio: 10 off in 7 w/days (then shipping): $130 ~ £103 (+ shipping)

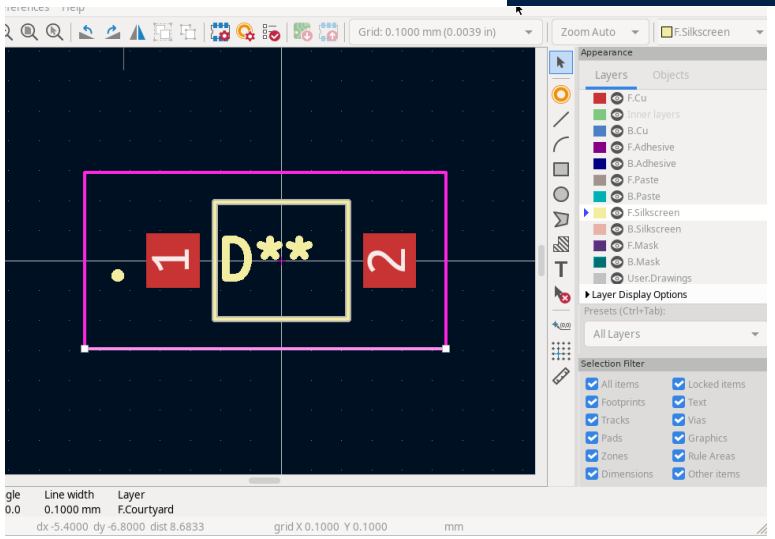


KiCad part creation example

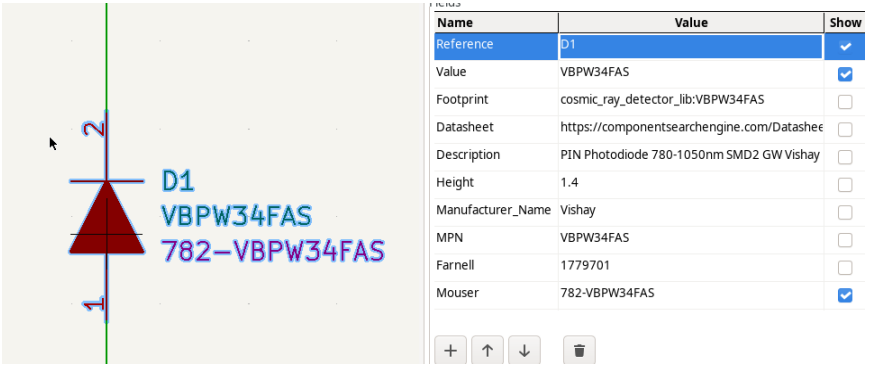
UNIVERSITY OF OXFORD

Drawing & Land pattern from datasheet (VBPW34FAS)





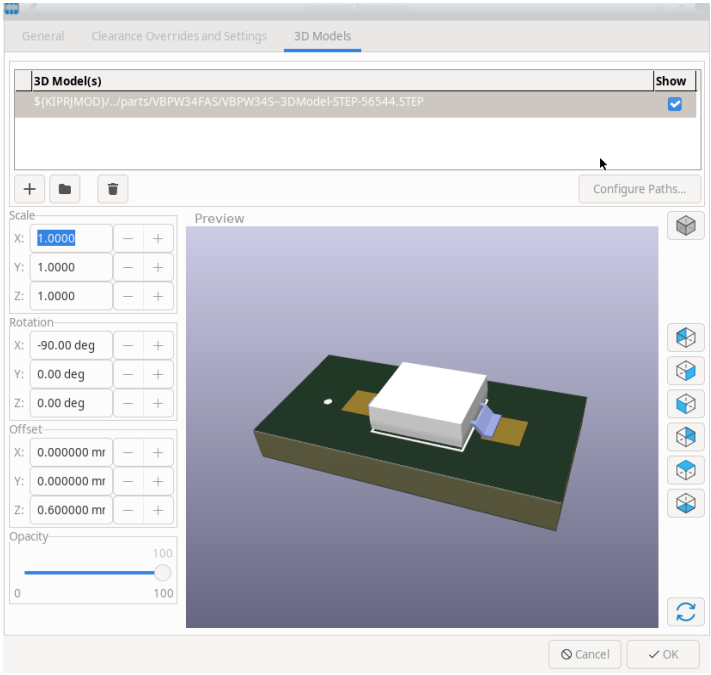
Below:
Use “generic” symbol,
And associate MPN,
Footprint and BOM data in schematic



Name	Value	Show
Reference	D1	✓
Value	VBPW34FAS	✓
Footprint	cosmic_ray_detector_lib:VBPW34FAS	
Datasheet	https://componentsearchengine.com/Datashee	
Description	PIN Photodiode 780-1050nm SMD2 GW Vishay	
Height	1.4	
Manufacturer_Name	Vishay	
MPN	VBPW34FAS	
Farnell	1779701	
Mouser	782-VBPW34FAS	✓

Above right:
Draw (or download) footprint and CHECK against datasheet

Below Right:
Download (or draw!!!) 3D model for MCAD integration



Kicad Schematic Tips

UNIVERSITY OF OXFORD

This applies not just to KiCad but schematic drawing for “real” PCBs in general.

- A schematic for an actual, real PCB that will be built is quite different from the “conceptual” schematic you jot down on paper or see in a textbook. The “little details” are important
- Any reasonably sized schematic doesn’t nicely fit on one page, and though it’s personal preference I like my schematic pages to not be too crowded.
- KiCad supports multi sheet design in two ways: **flat** multi-page designs with global labels, and **hierarchical** designs with hierarchical labels. I prefer this latter and again that’s personal preference but for the purposes of today this is how we will look at it.
- Associate as much data with the schematic before layout as possible, this includes: footprints, BOM data, datasheets and netclasses.
- Just like writing a computer program, put **text** comments on your schematic to indicate intent.
- Make sure to **label** important nets with useful names so you can easily see them whilst in layout

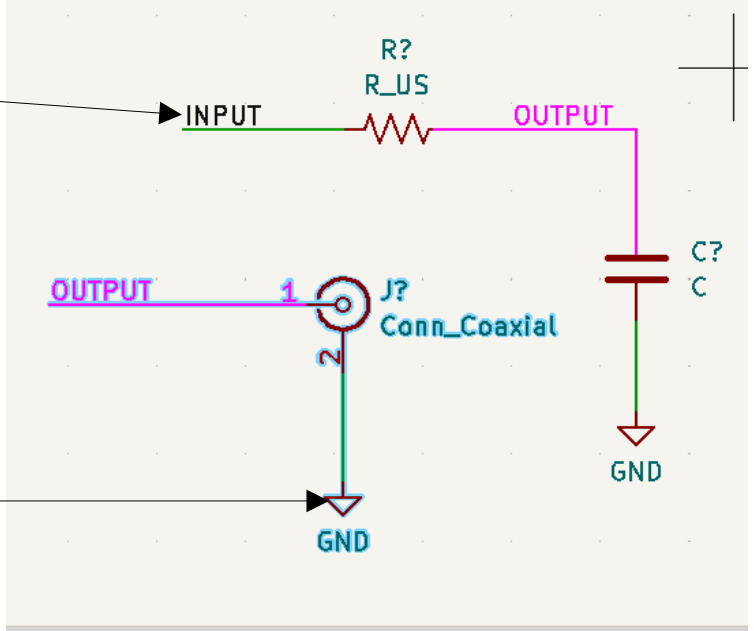
Schematic Labels

Normal labels: the two nets labelled "OUTPUT" are **electrically connected**

(this is common in most modern PCB Software, simplifies schematic spaghetti).

But, ONLY if they are on the same sheet!

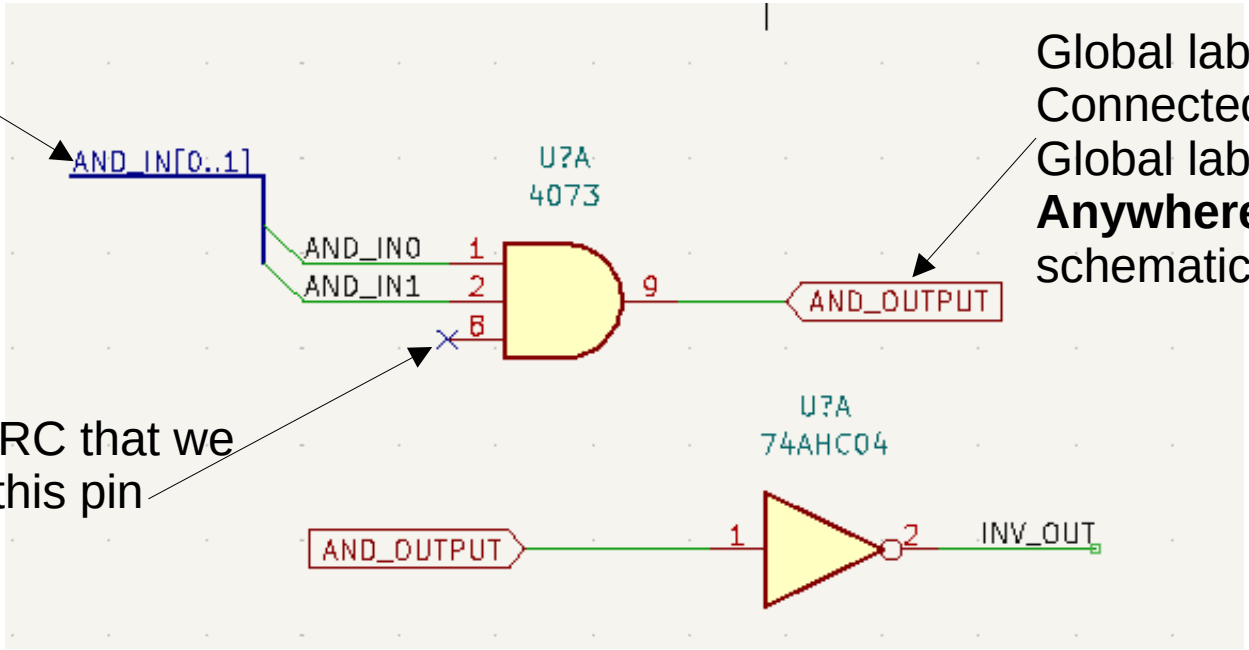
ALL power symbols (e.g. "GND") with The same name are connected **globally** Throughout the schematic.



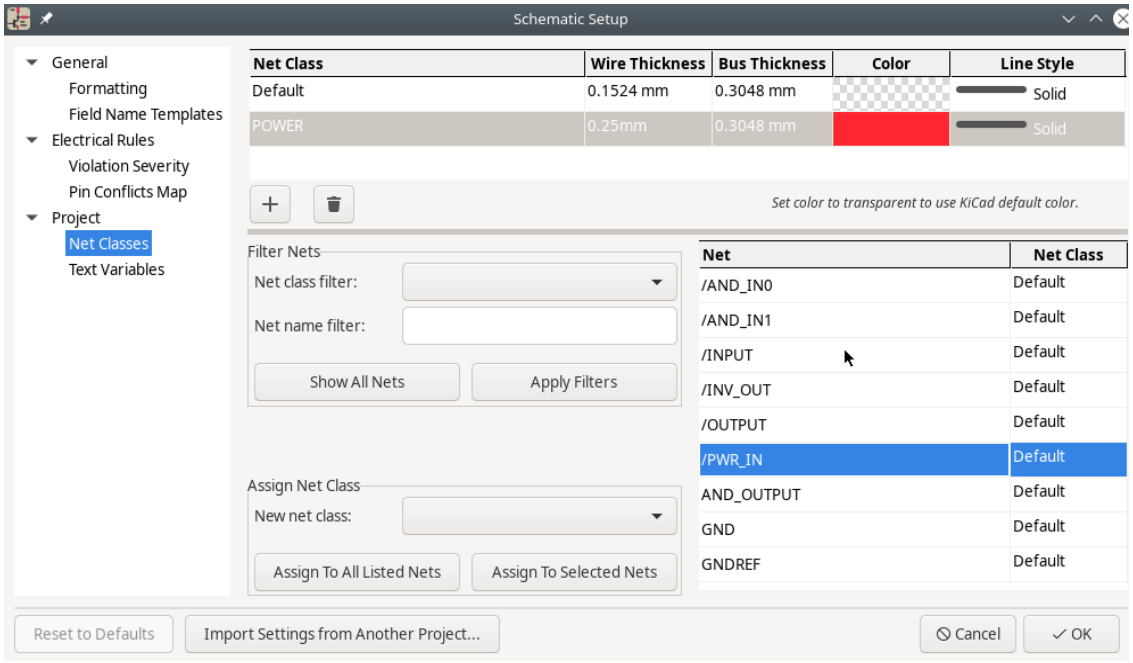
Buses – groups of similar signals

Global labels - Connected to Global labels of same name **Anywhere** in a multi-page schematic`

NC symbol - Indicates to the ERC that we Will not be using this pin

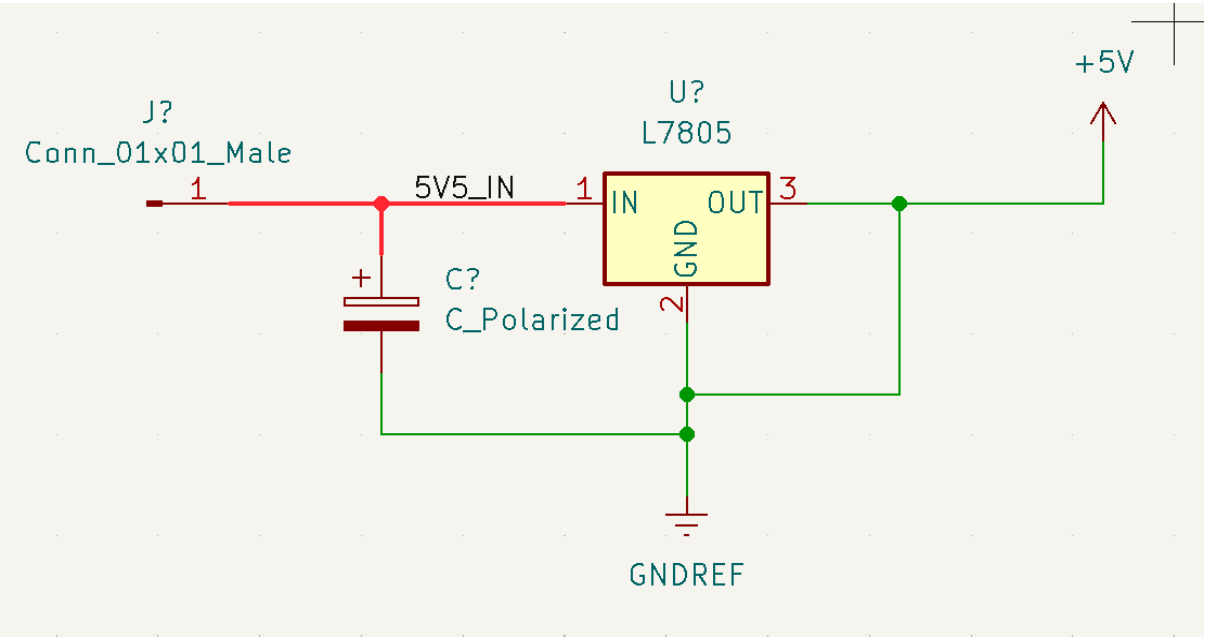


Netclasses

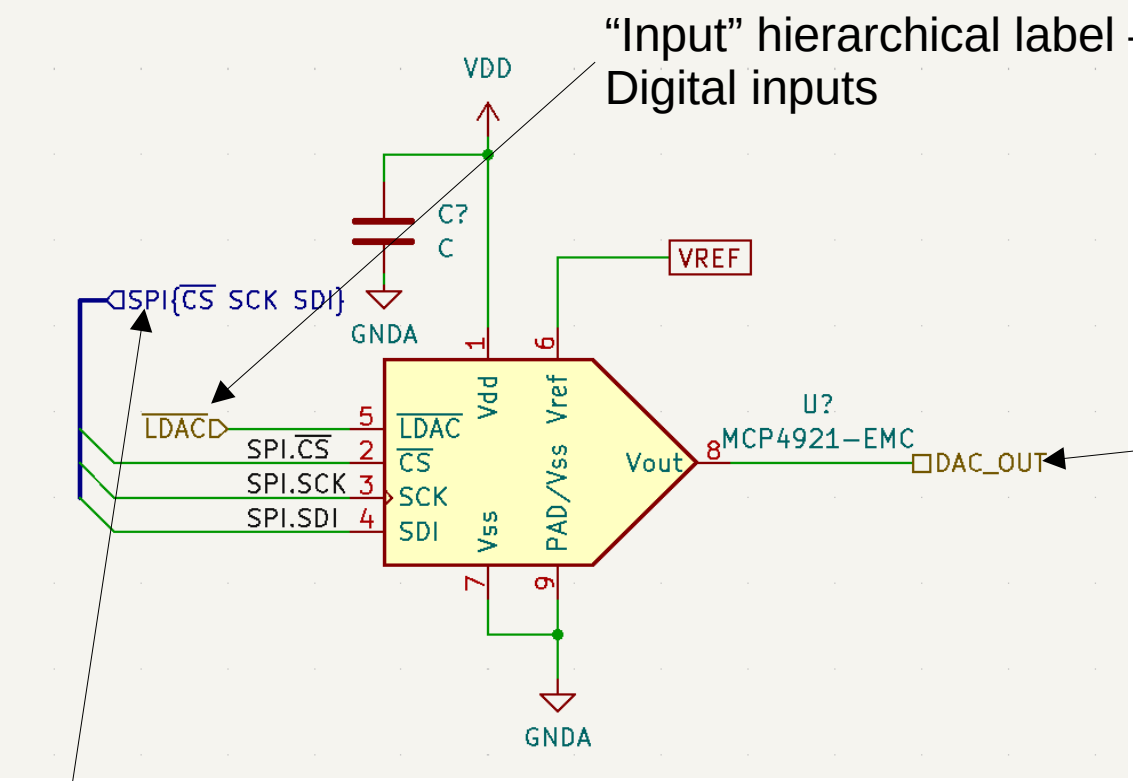


Netclasses are just “types” of net, that we might want later for PCB routing purposes (e.g. to make all power input nets have thicker and more separated traces). We set them up in the “Schematic Setup” dialog

Right: We have assigned the “5V5_IN” net to the POWER netclass, this gets transferred to the PCB routing



Hierarchical Schematics



"Input" hierarchical label – for Digital inputs

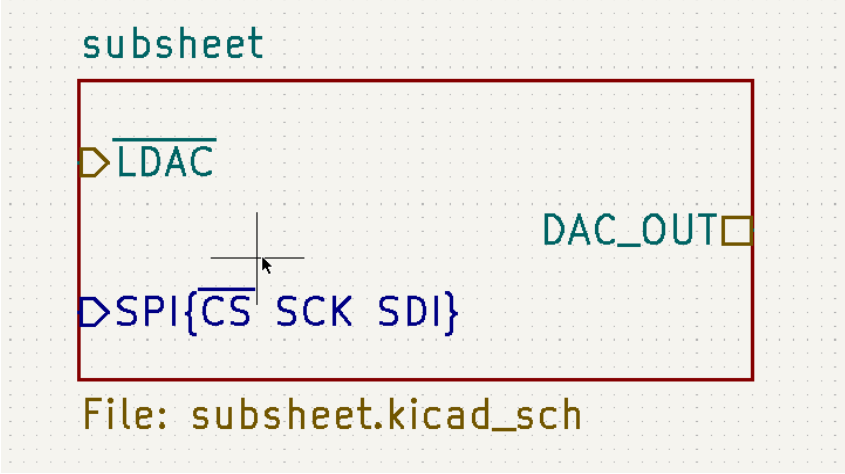
Hierarchical Labels – connect Through sheet symbols. (left) – the contents of "subsheet.kicad_sch".

"Passive" hierarchical label - use for all analog signals

Below: the top level sheet. Each hierarchical label in the subsheet appears here. We can connect wires and buses to them as though the sheet were a component

Group Bus – groupings of named Nets inside a bus (but not numbered) – useful for well defined interfaces e.g. SPI, I2C, UART etc

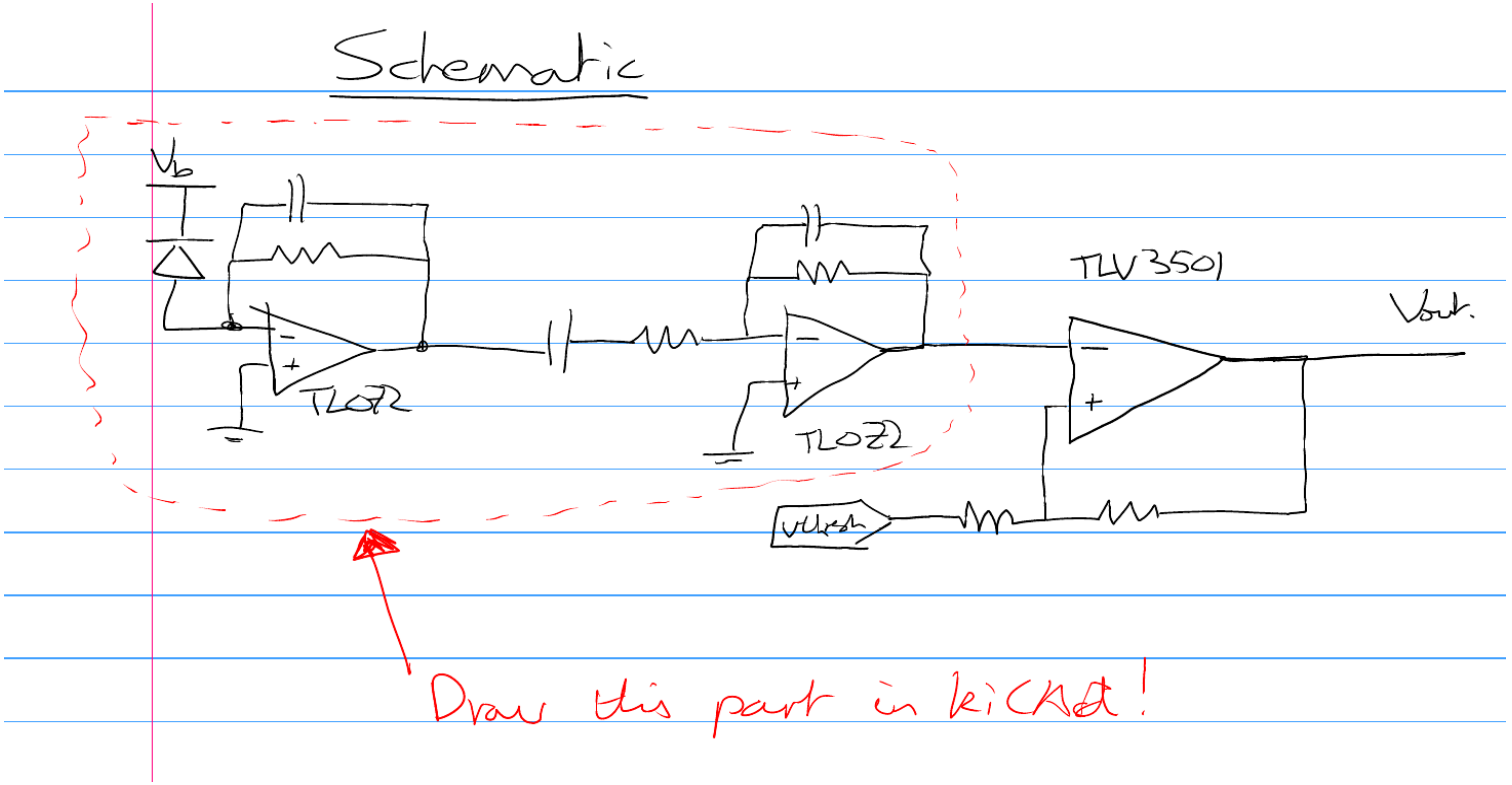
Hierarchies can be as deep as needed. You can also use the same schematic File in multiple sheet symbols if you need "suplicated" subsystems in the design



Walkthrough KiCad Schematic

On the Indico page, you will find two .zip files:
1) a KiCAD project for a “PIN diode based gamma ray detector” which has a pretty complete schematic, including data for parts, MPNs, footprints etc “schem_proj_complete.zip”

2) the same project but with a crucial part of the schematic missing – the frontend for the detector! The challenge is to draw in the missing part of the schematic. “schem_proj_nofrontend.zip”
We will talk through how to do this together to learn some of the KiCad basics.



Schematic Steps Overview

UNIVERSITY OF OXFORD

1)Draw basic circuit – the symbols should all already be included in the project in the “cosmic_ray_symbols” library

2)Label & connect – add suitable net labels and connect the new sheet into the hierarchy properly

3)Annotate – annotate the schematic (assign specific designators to each part)

4)Pass ERC – run the ERC and tweak until there are no more errors!

5)Assign Footprints & add BOM data - associate each schematic symbol to a physical footprint and part to order

6)Assign netclasses – We will assign some netclasses for traces like “power” and “bias” to associate custom design rules with them at layout time

7)Set up sheet block and export drawings for review

Thanks!

UNIVERSITY OF OXFORD

To be continued on Friday where we get to the fun part – layout!

Questions, comments etc greatfully received:

Daniel.weatherill@physics.ox.ac.uk

This lecture is brand new for the first AITL series, all feedback is VERY USEFUL!

Very happy to be contacted about “stupid questions” re: KiCad or any PCB designs. I’ve done somewhere around 30 over the years now (in KiCad, altium, Eagle, DipTrace- urrrgh) so do know a little bit about what I’m doing

UNIVERSITY OF OXFORD

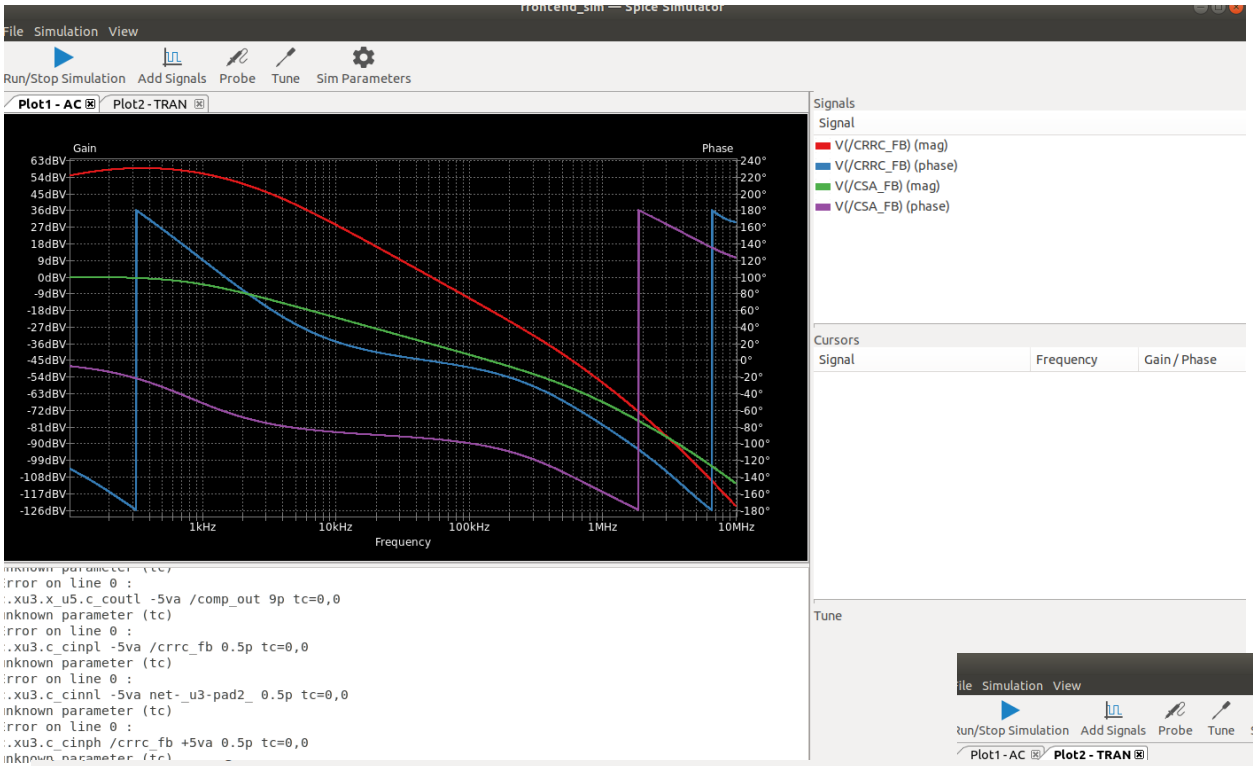
Advanced Instrumentation Lectures: PCB Design with KiCad Session #2

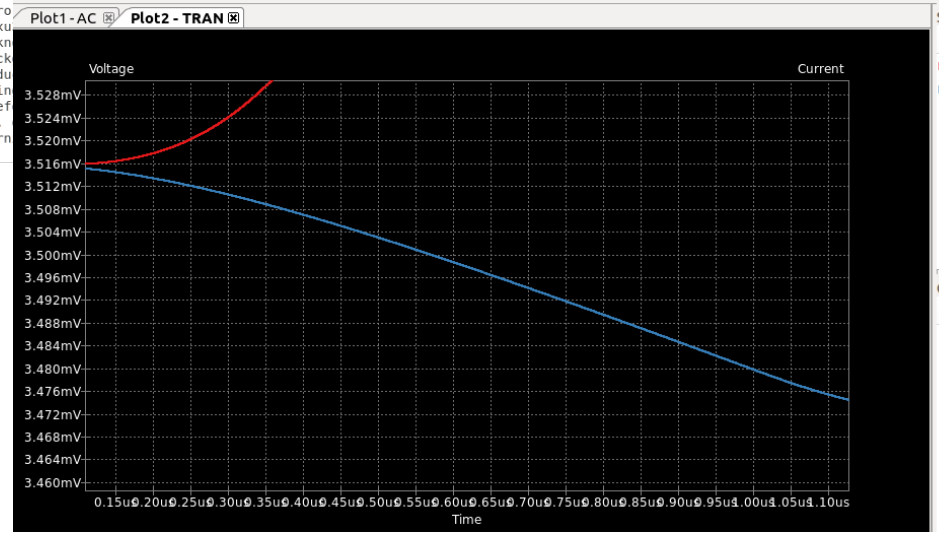
Dan Weatherill

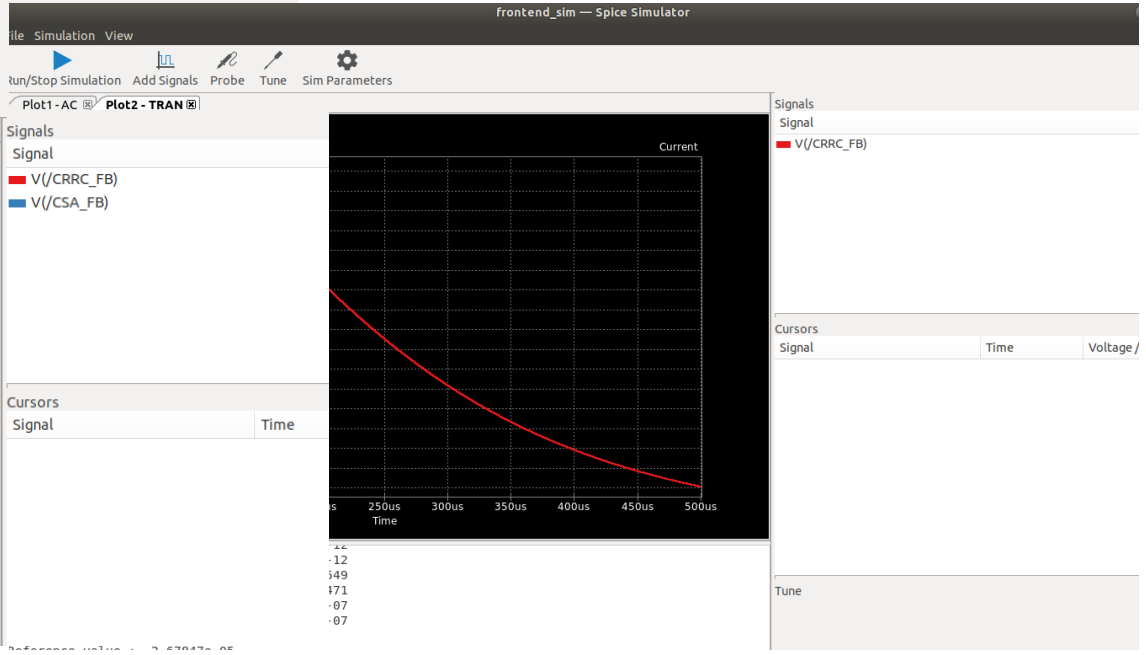
Oxford Physics Microstructure Detector Laboratory

Kicad SPICE simulation example

UNIVERSITY OF OXFORD





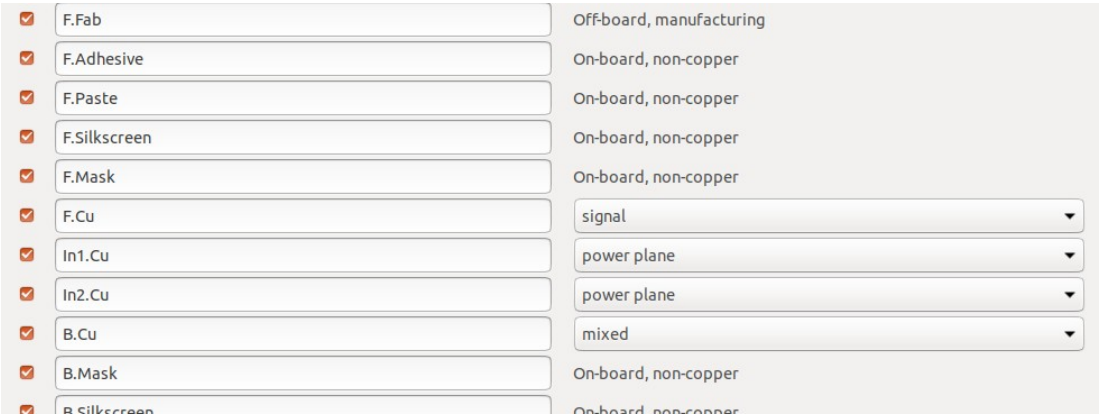


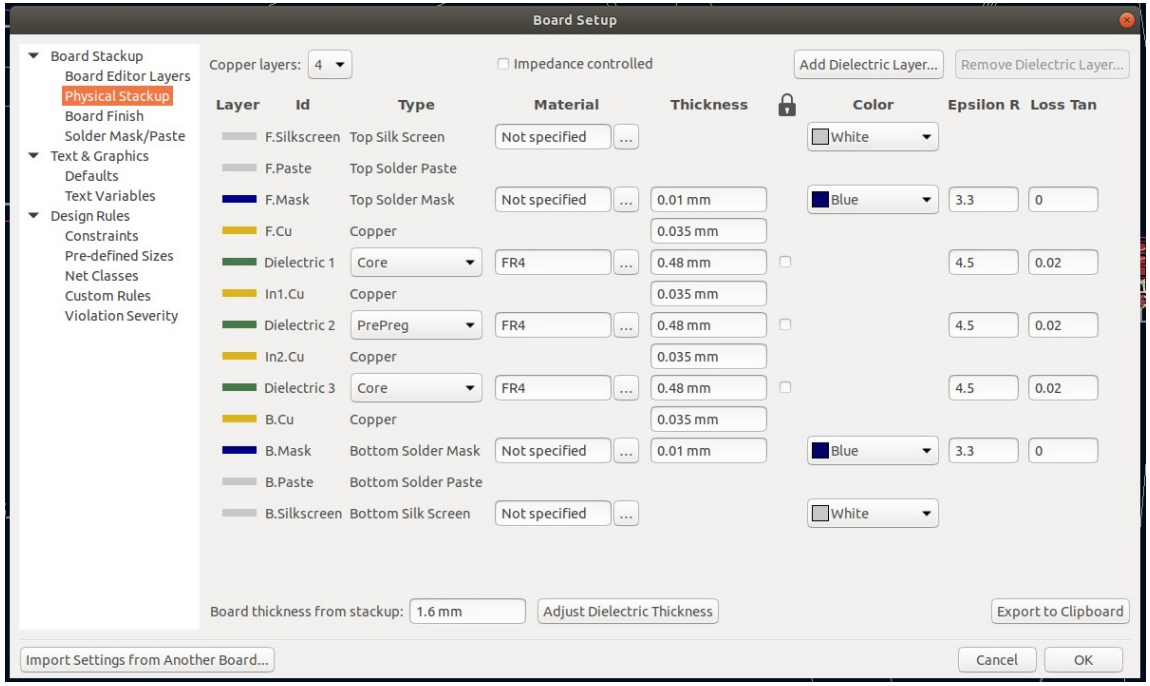
Built in access to the ngspice simulator, which is quite nice (though not effort free!). Ability to probe and adjust values directly from a KiCad schematic.

No time to go over in detail today unless someone really wants a demo, but example included on Indico

KiCAD PCB layout - layers

UNIVERSITY OF OXFORD

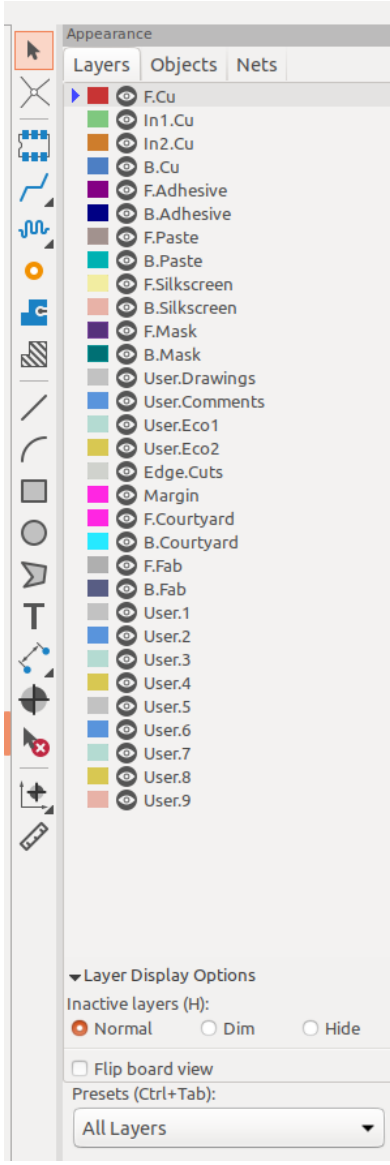




Be sure to setup layer names, intent and stackup thicknesses in the “board setup” dialog.

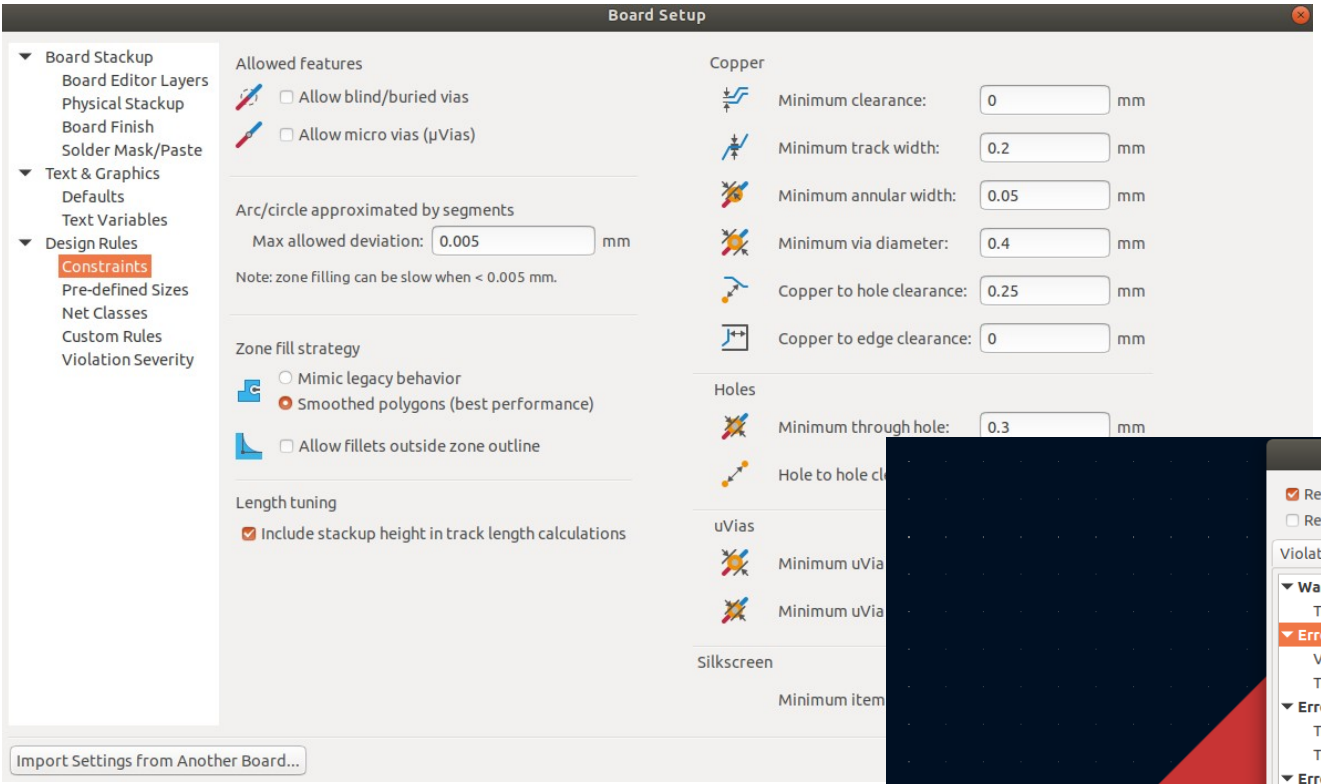
(currently impedance settings don’t do much beyond add to export data and adjust 3D view, “true” controlled impedance in next version!)

(**RIGHT)**: lots of display options available, tweak to however you need

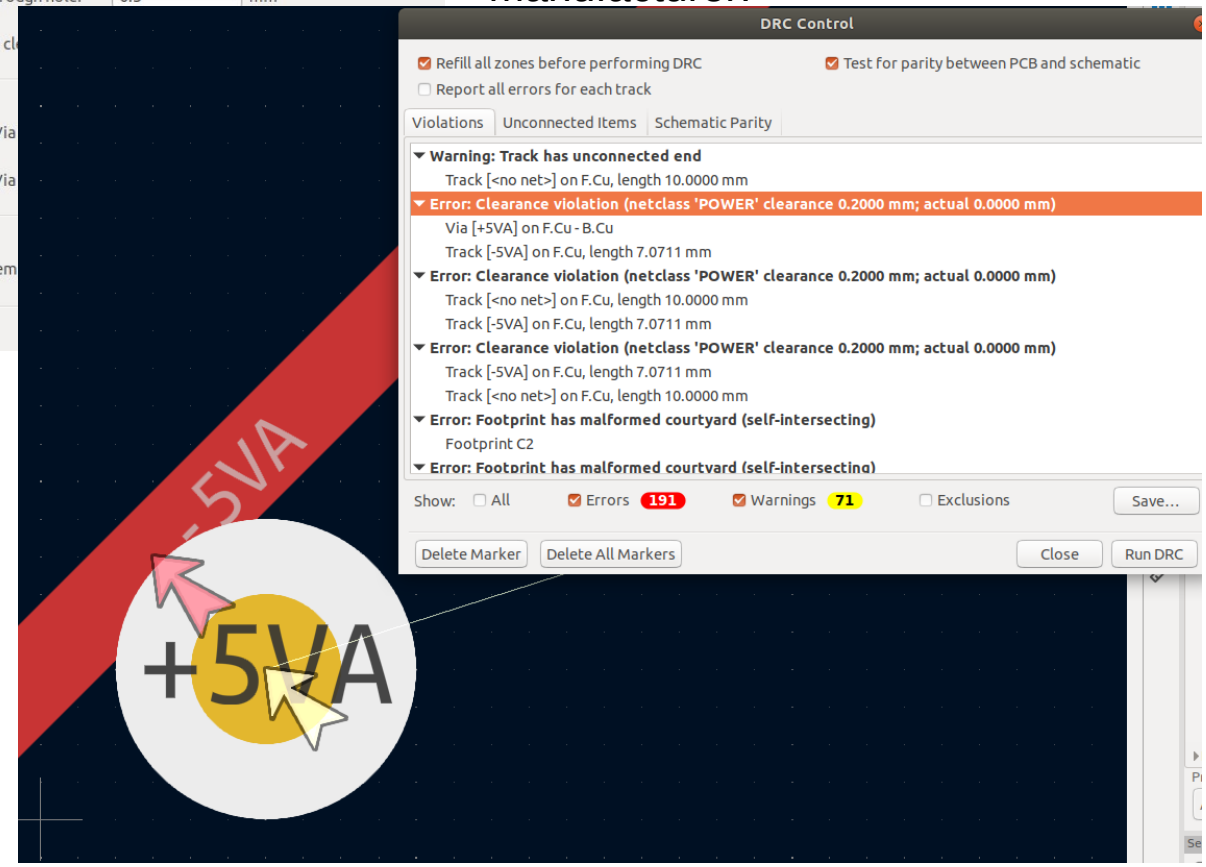


KiCAD PCB layout – design rules

UNIVERSITY OF OXFORD



Set up your design rules according to your manufacturer's stated capabilities (again in "board setup" dialog). You can import DRC settings from a previous project – so e.g. keep a "template" project for each manufacturer.

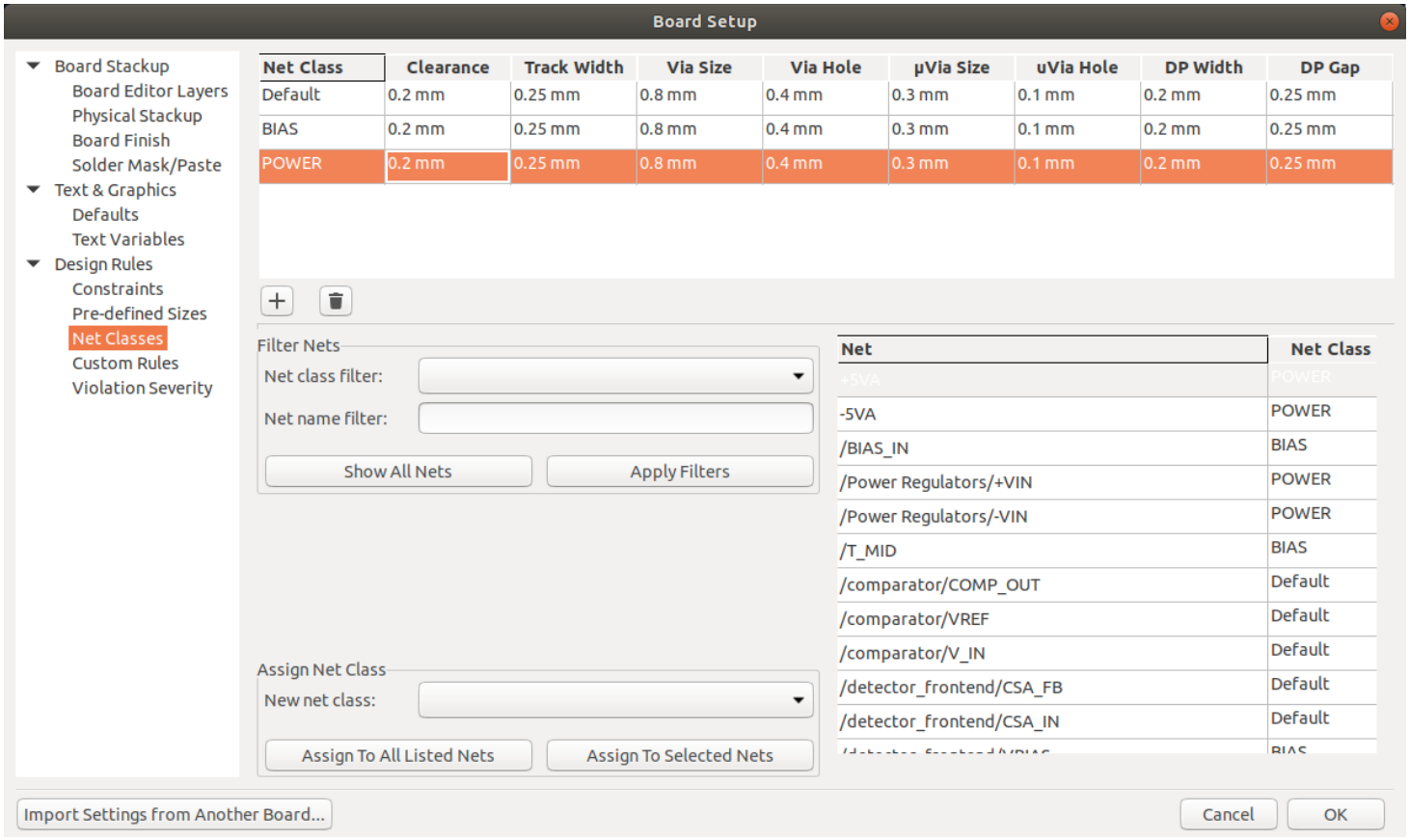


Set up DRC **before** doing any routing to avoid disappointment!

Using the minimum allowed is not always best strategy!

The interactive router does generally prevent you from violating clearance but you can force it to, and hit it at DRC time (right)

KiCAD – netclass rules



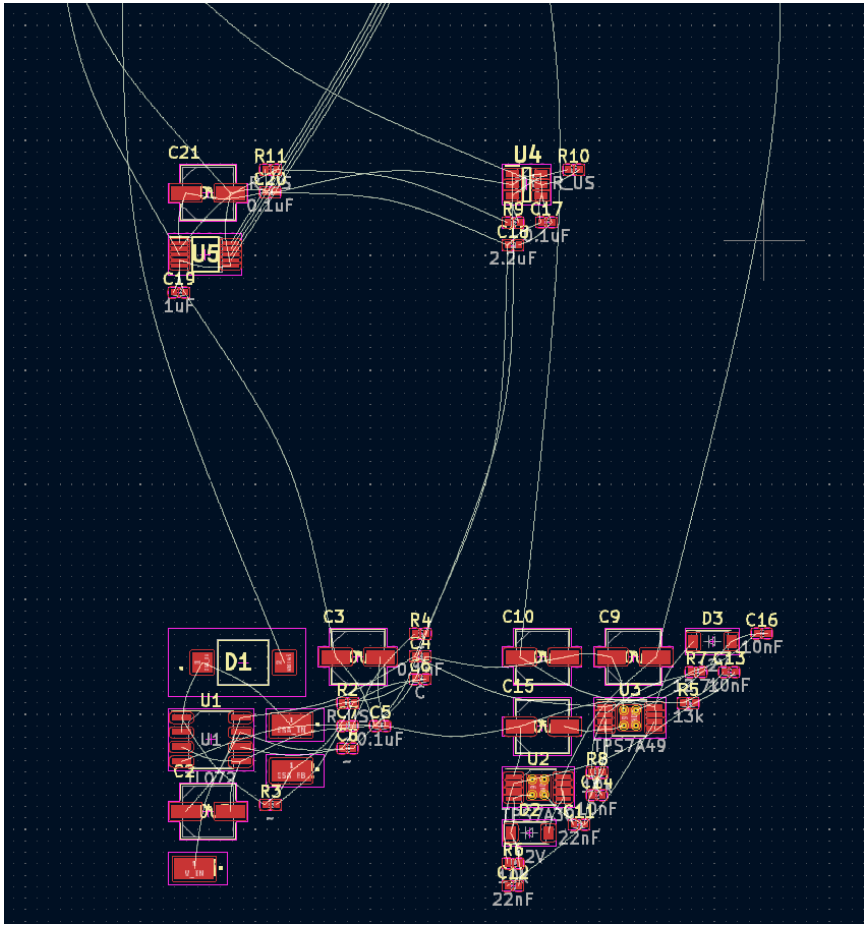
Netclasses you created in the schematic appear also in “board setup”, and can have different sets of design constraints applied to them (**left** we have power traces with higher thickness and clearance than default)

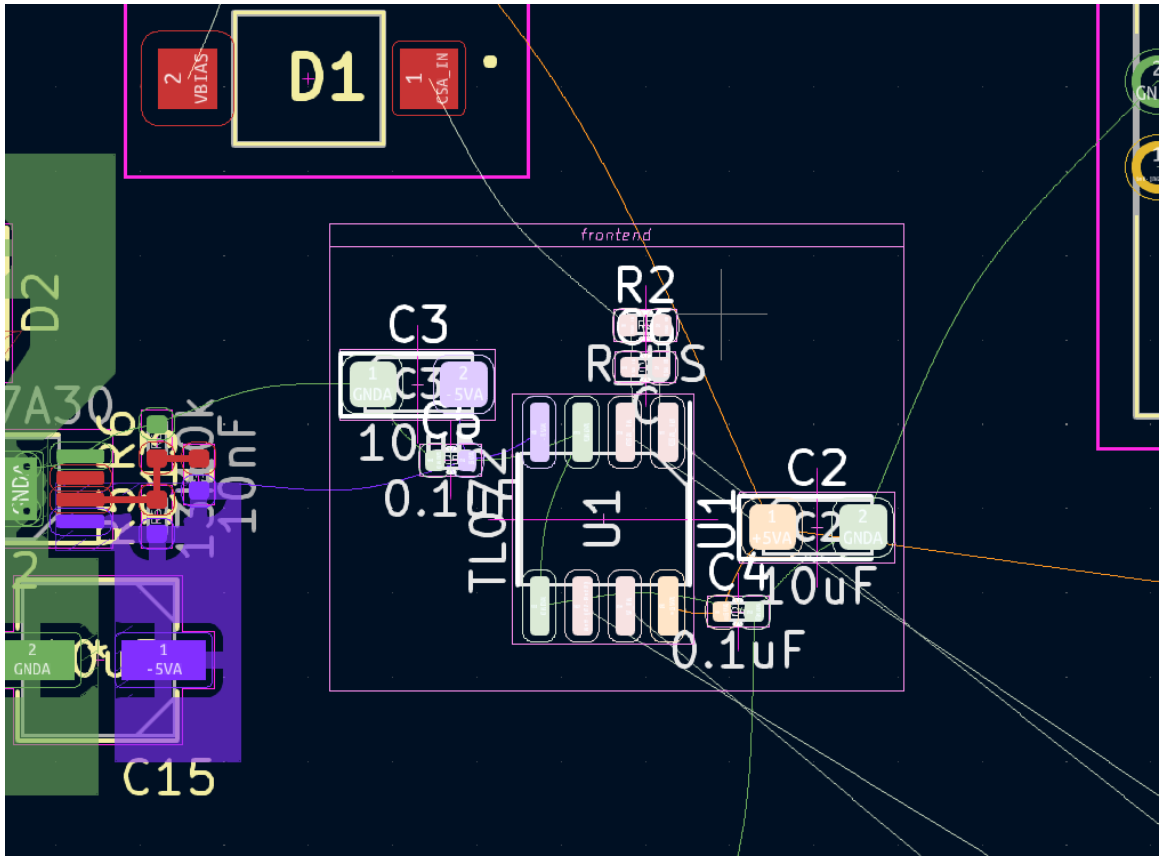
In fact, you can actually write completely custom DRC rules to suit many needs – **right** Newbury has a different minimum hole clearance on inner layers than outer ones, this isn’t available in the default setup but a simple rule can be written to enforce it!

DRC rules:

```
(version 1)
(rule inner_layer_hole_track
    (layer inner)
    (constraint hole_clearance (min 0.5mm))
)
```

General PCB layout -ratsnest





Generally, best to start by getting components placed reasonably well before drawing tracks.
The “ratsnest” is a good guide for placement and rotation

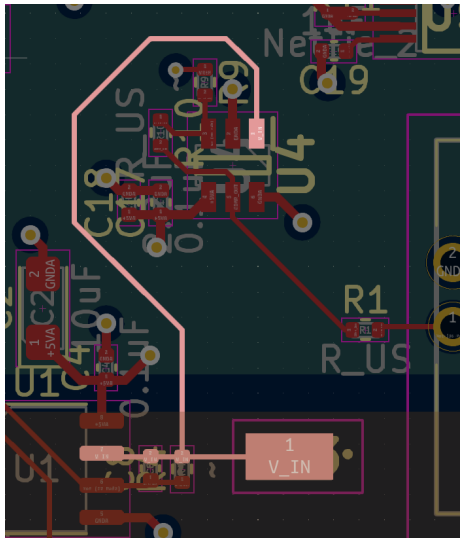
In some circumstances it makes sense to “group” a block of layout so it can be moved freely around and piece together the whole layout later. Be careful, though, this can often lead to unoptimised layout

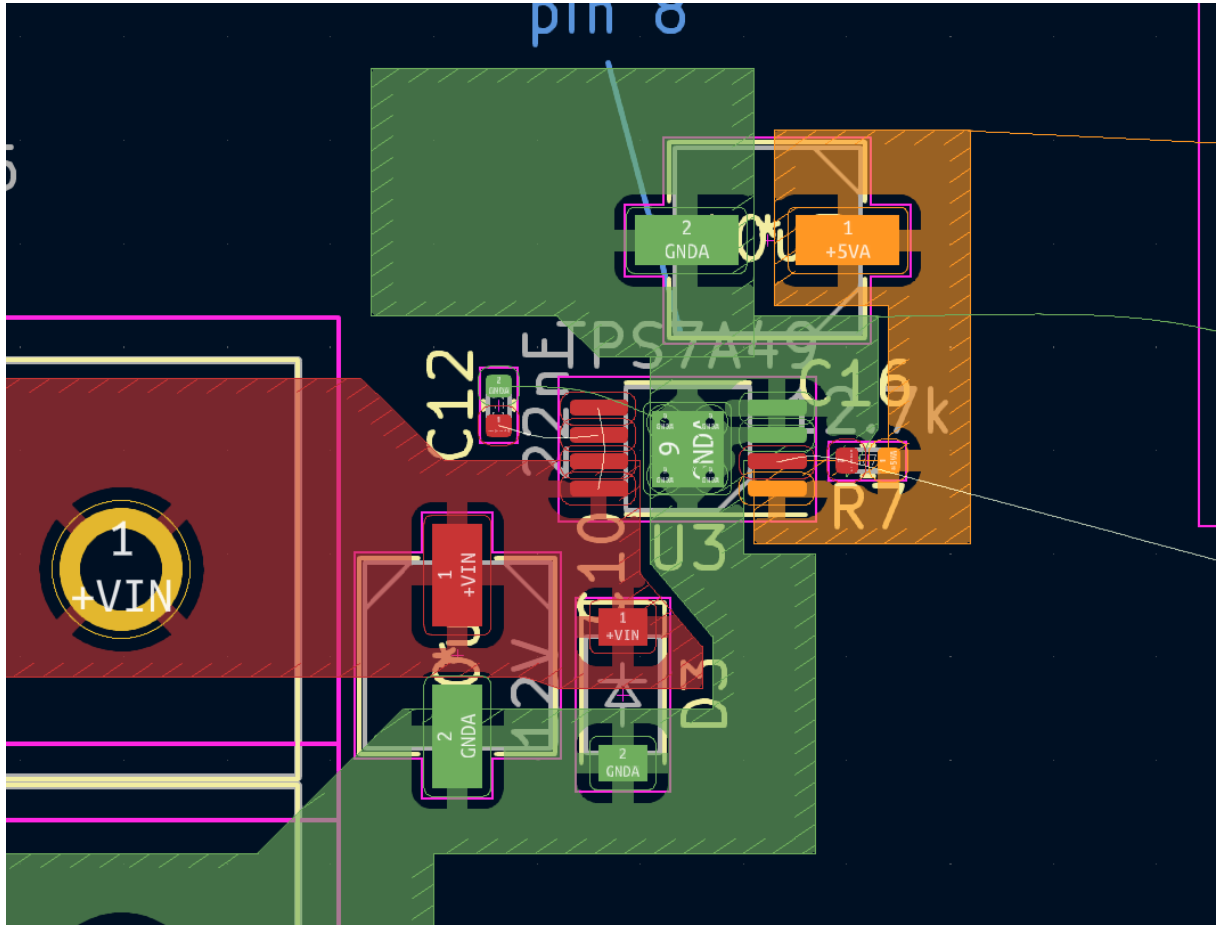
General PCB Layout – zones/pours

UNIVERSITY OF OXFORD

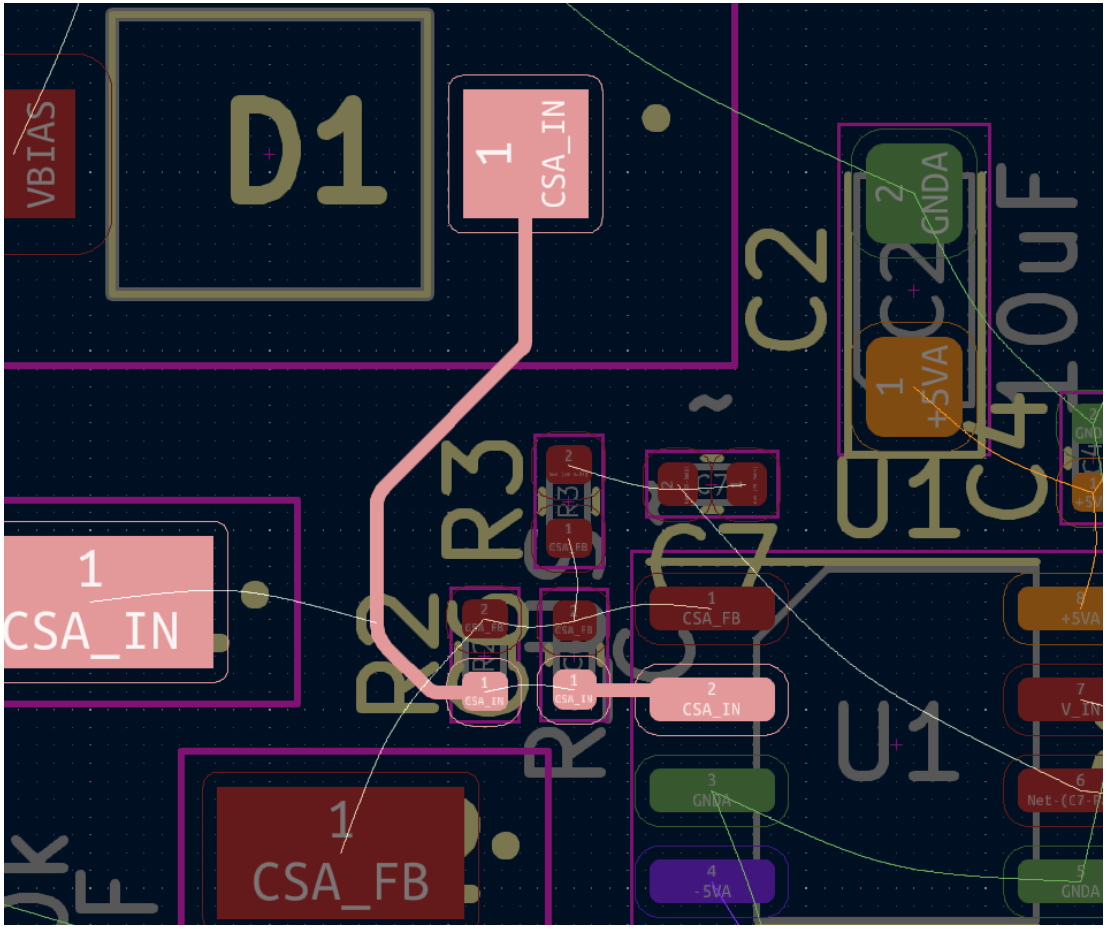
Always follow instructions for layout in component datasheets carefully, particularly for anything that deals with power. For modern SMD designs you will likely want quite a few “pours”, large areas covered in copper. Be sure to tweak the parameters how you want them (e.g. make sure to have thermal relief rather than solid connections in big zones or soldering them will be a nightmare). In addition, try to fill up empty board space with copper pours and even out the copper mass on both sides of the PCB: this will prevent warping over time

When using PCB software, take advantage of the ability to adjust the display – e.g. on the **right** I have set the colouring to be by net rather than by layer (which is the default). **Below** we have set the display to only some layers and highlighted an important net we are working on



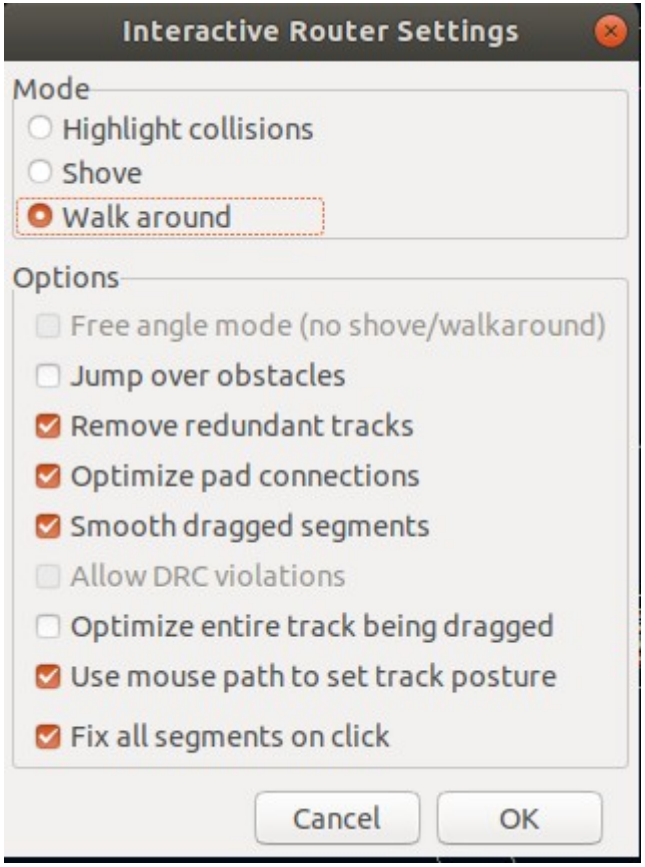


KiCAD Layout – Routing



The interactive router has a few settings you will be changing often.

Getting to know keyboard shortcuts is a great time saver!
Generally, route important tracks first, or those with electrical constraints (impedance, length etc)



The default (and probably most useful) mode is “walk around”, which routes tracks according to the design rules by optimising their geometry. The “shove” mode allows the track you’re currently routing to push others out of the way – useful for priority tracks.

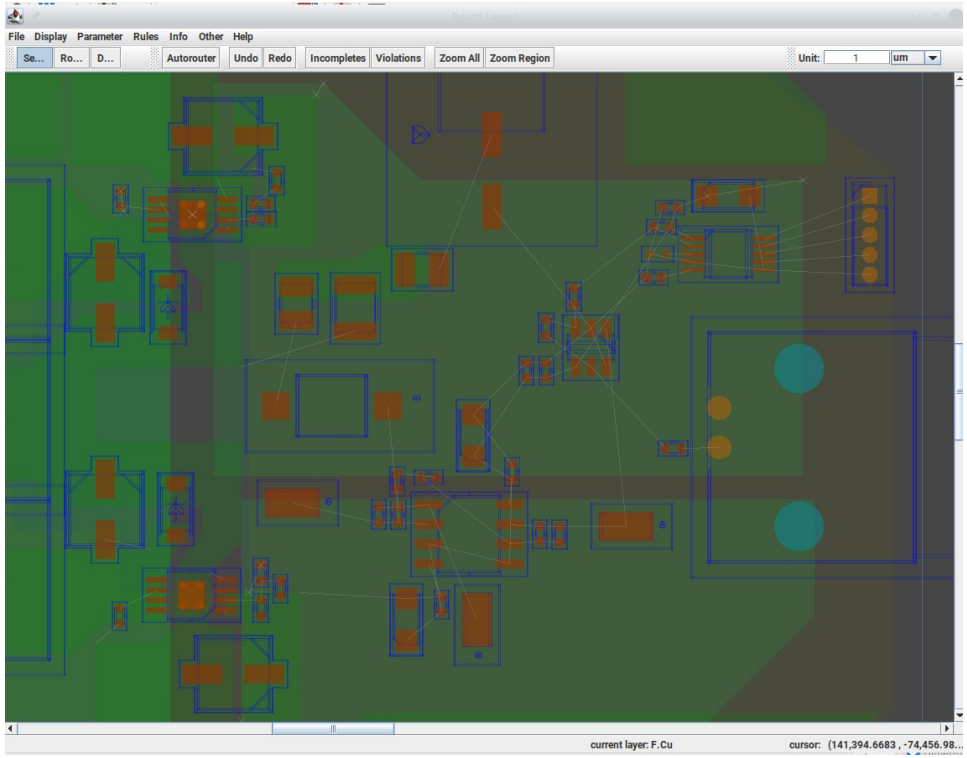
Occasionally you need the “highlight collisions” mode, which allows you to break the DRC and just let you know you are doing so

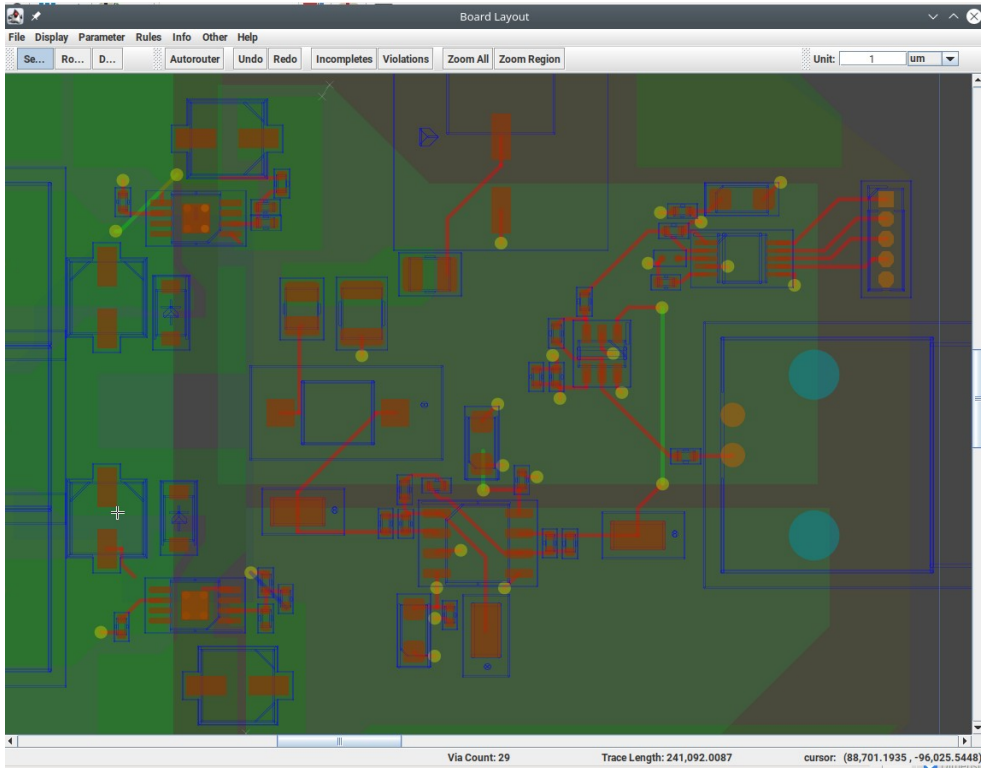
Autorouting

UNIVERSITY OF OXFORD

Kicad has no built in auto-router, but it can export the Specctra “DSN” file, to use with external tools. One such is freerouting:
https://github.com/freerouting/freerouting

It isn’t that bad! **BUT:** autorouting generally only produces good results if you’ve already done a nice job with placing components and placing zones for ground, power planes etc. Even then, you will mostly want to “tweak” the results quite significantly. **Below:** exporting a KiCad design to freerouting, and 1 click autorouting it. The result can be re-imported back to KiCad





KiCAD routing - DRC

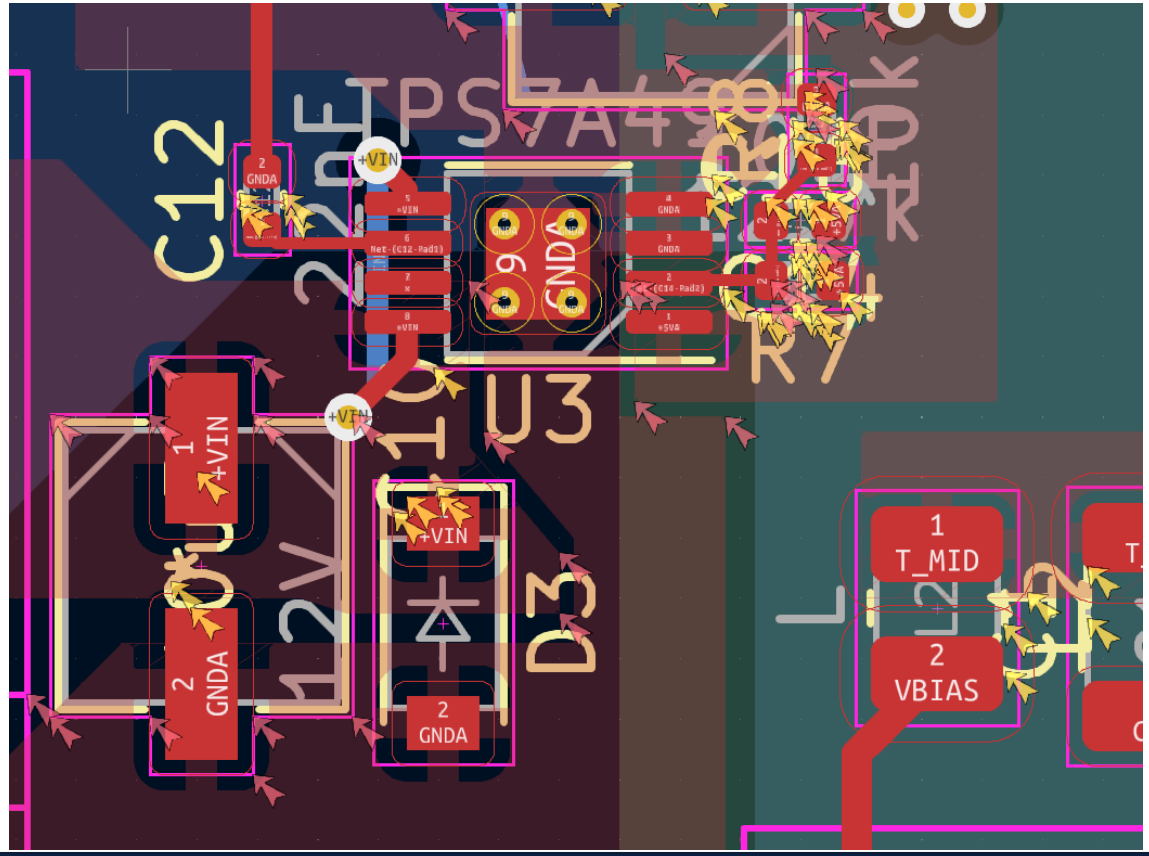
Getting to “0 DRC” can be daunting but it is mostly worth it!

Look through the errors, many are real and useful!

To get rid of the rest, you can:

- Adjust library symbols so that the clearances are improved
- Add specific exclusions for the times you **know** you meant to break the rules (e.g. the overlapping connectors on the cosmic ray detector board)

  Note that KiCad’s DRC isn’t quite as smart as Altium’s – one misplaced component can create a LOT of arrows!

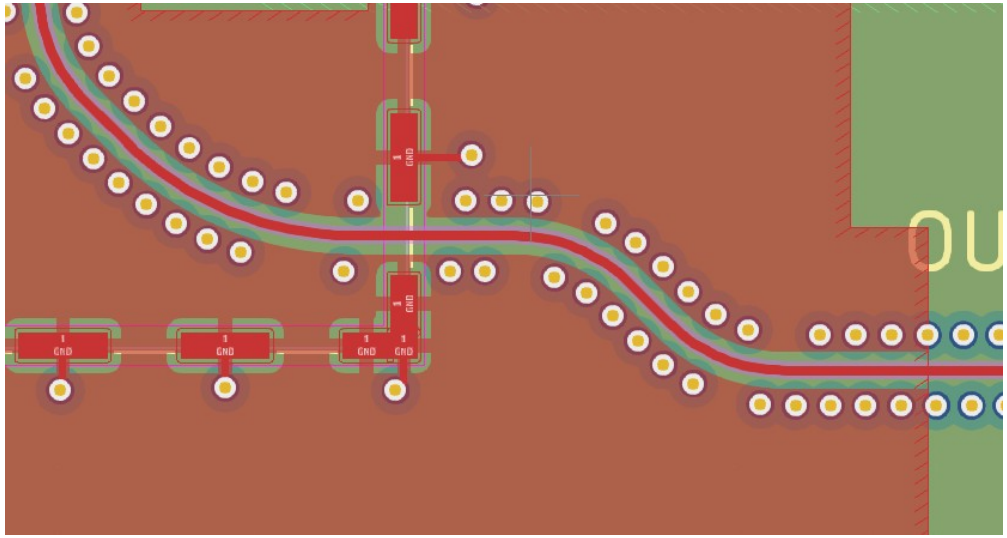


KiCad external plugin – via fence

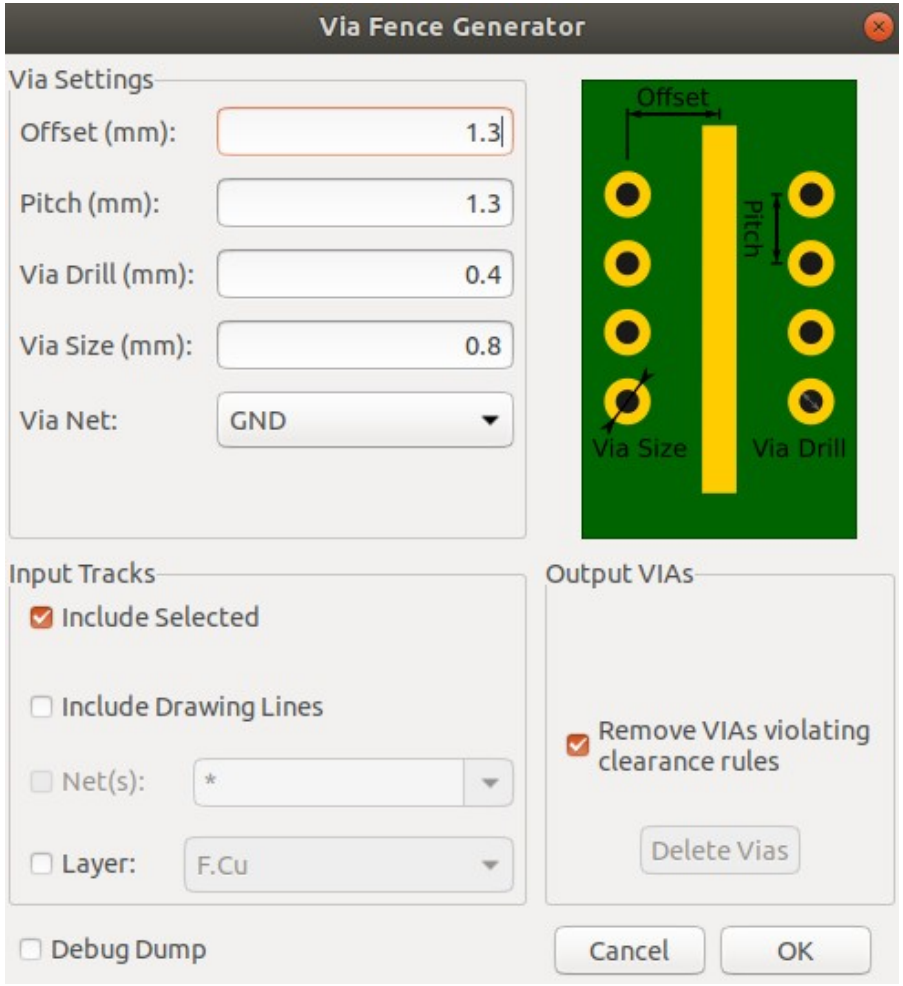
https://github.com/easyw/RF-tools-KiCAD

Excellent plugin (though at moment has to be manually installed, not available through the built in “plugin” menu), that contains several tools useful for high frequency/RF design, notably the **below** via fenced and mask removed traces.

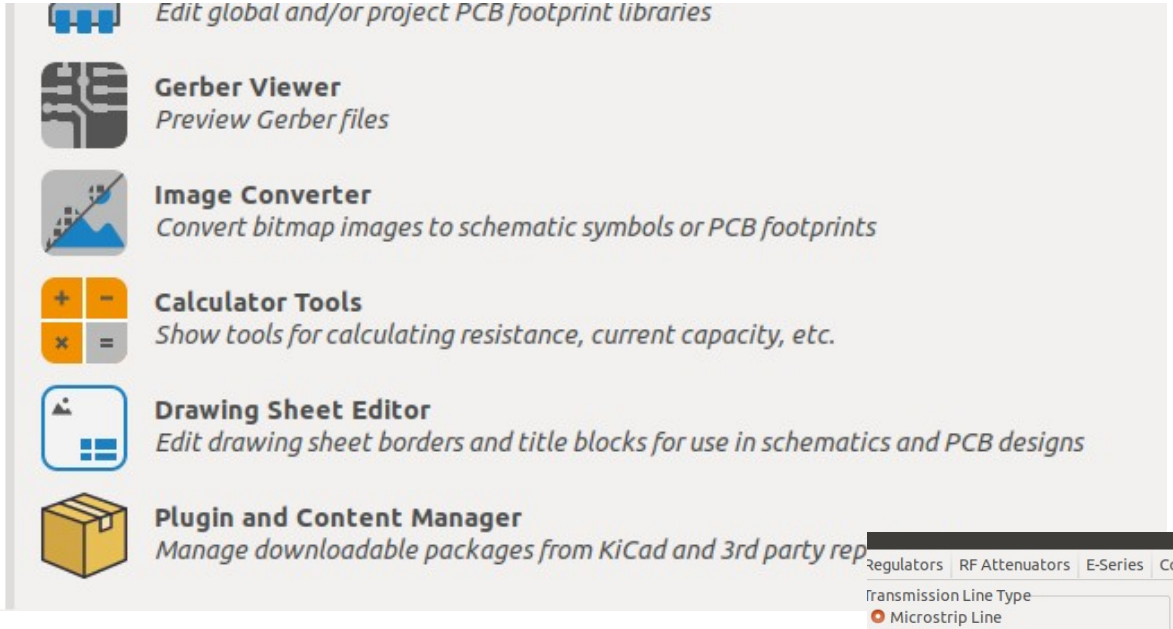
NB the below design is for an LGAD amplifier that should work up to ~10GHz. So far we’ve only tested it to about ~4GHz, but that is another story.



WARNING: not fully compatible
With kicad 7.x yet! (can be achieved
Manually now, but ask if interested!)



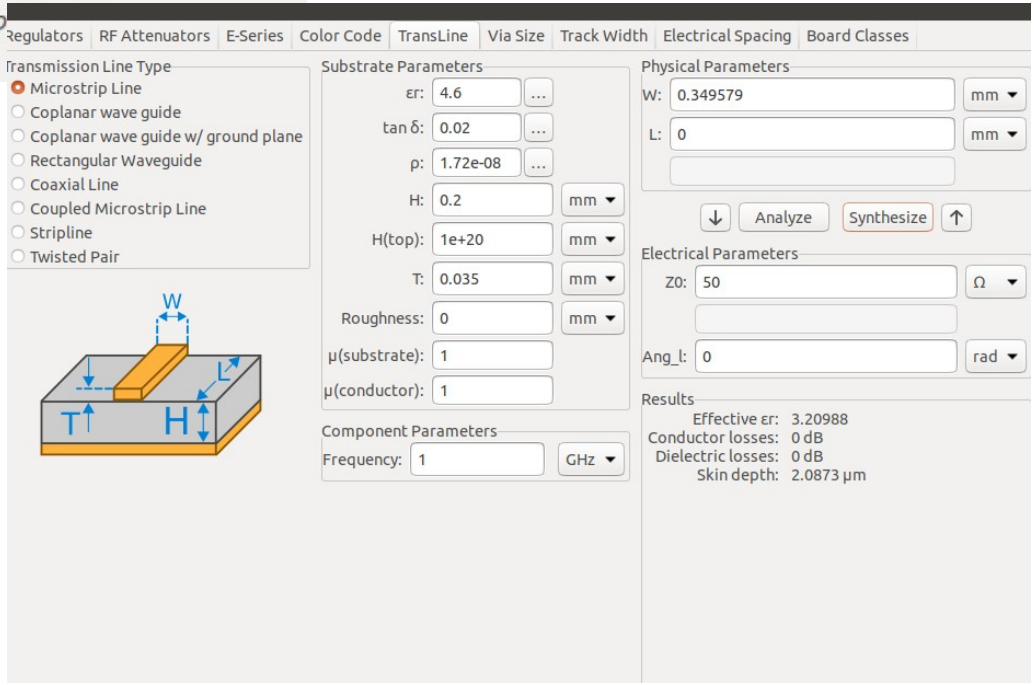
Trace thickness calculation



Lots of useful utilities hidden towards the bottom of the initial KiCad project window!

Though we don't (yet) have built-in support for calculating controlled impedance traces, we DO have this handy calculator that can synthesize length & width parameters after inputting board information (under "PCB calculator, TransLine").

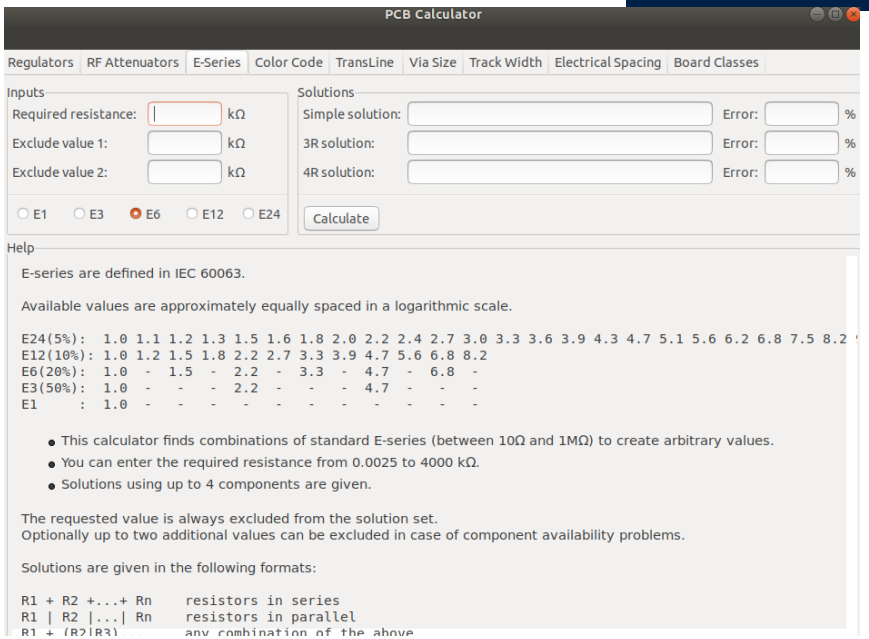
We also **do** have built in length tuning in the PCB editor (ask for demo if I forget!) but you have to know the length you want in advance.

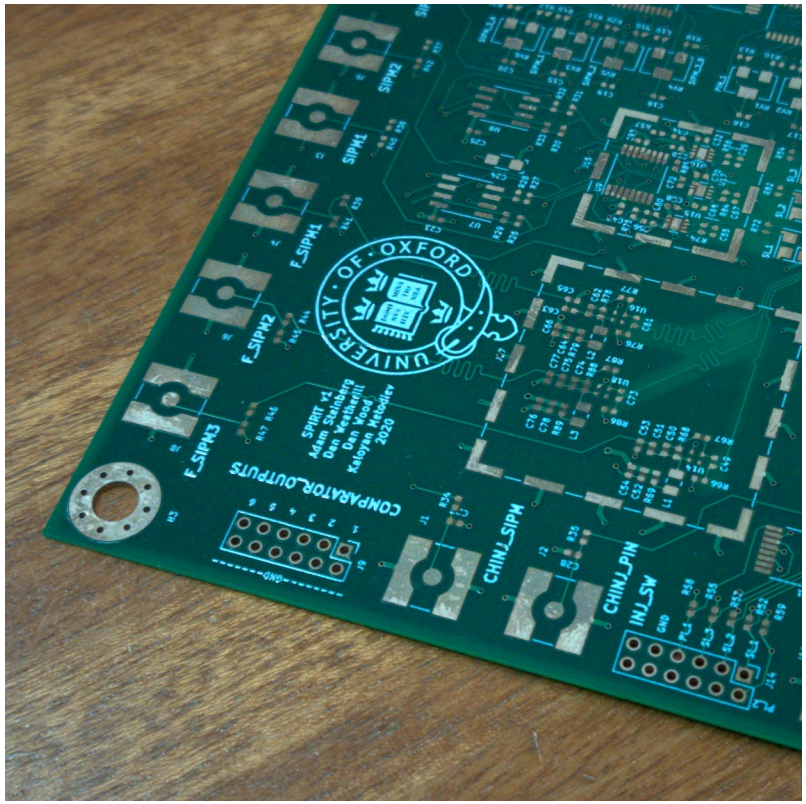


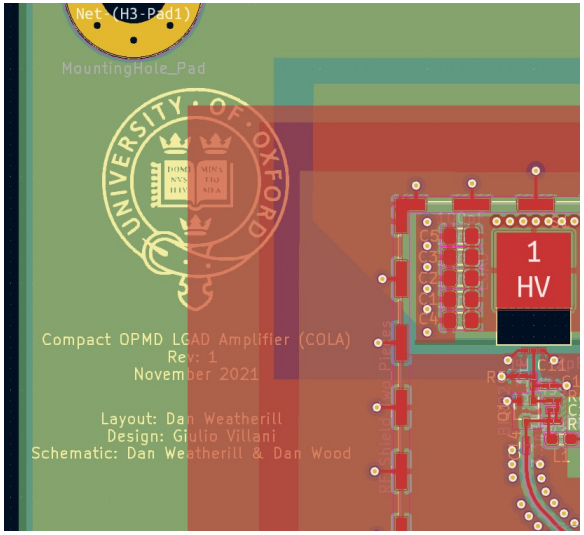
Other Useful Utilities

Other very useful tools towards the bottom of the kiCAD project menu:

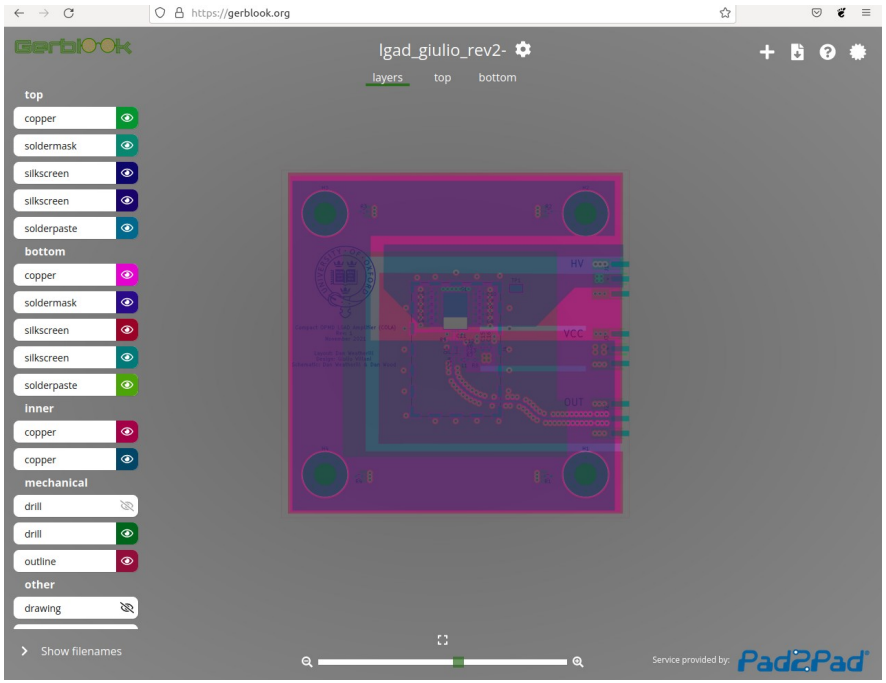
- In “PCB calculator” the E-series calculator, tells you what E series resistors you need to make a particular value
- The “Image Converter” utility converts (black & white) PNG images into PCB footprints. That is how we can do e.g. logos (**below**)

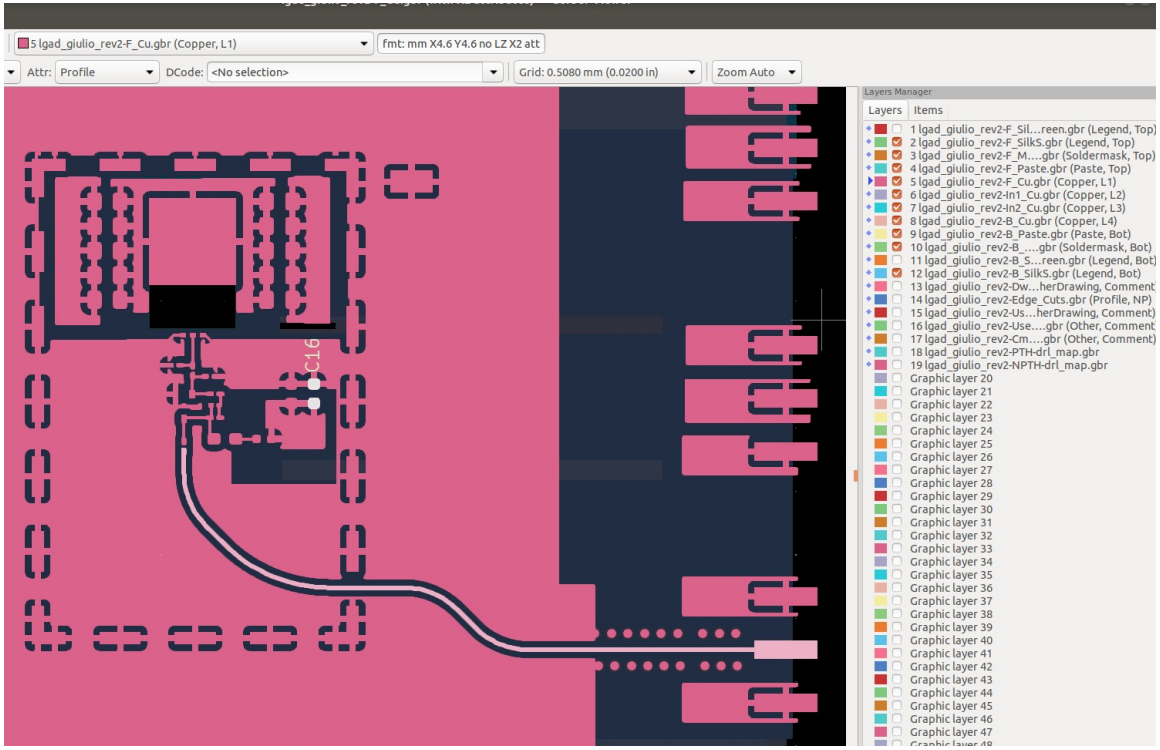


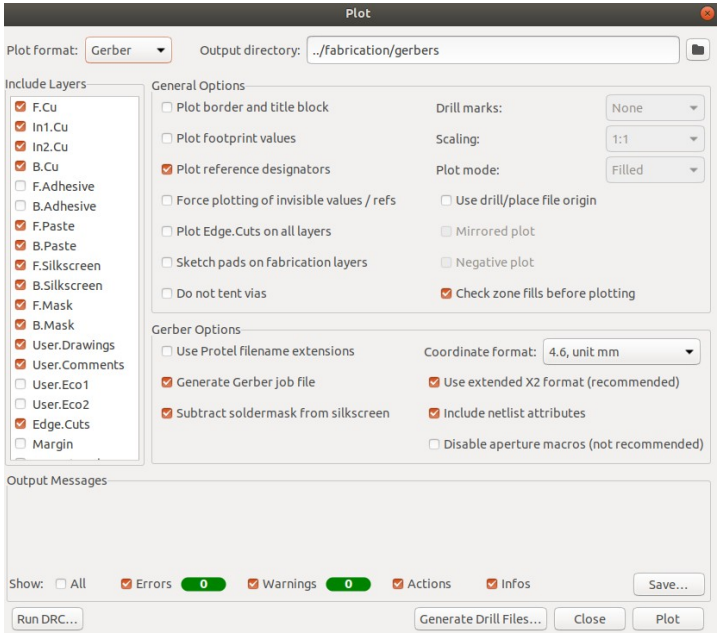




KiCAD – Gerber & Excellion export





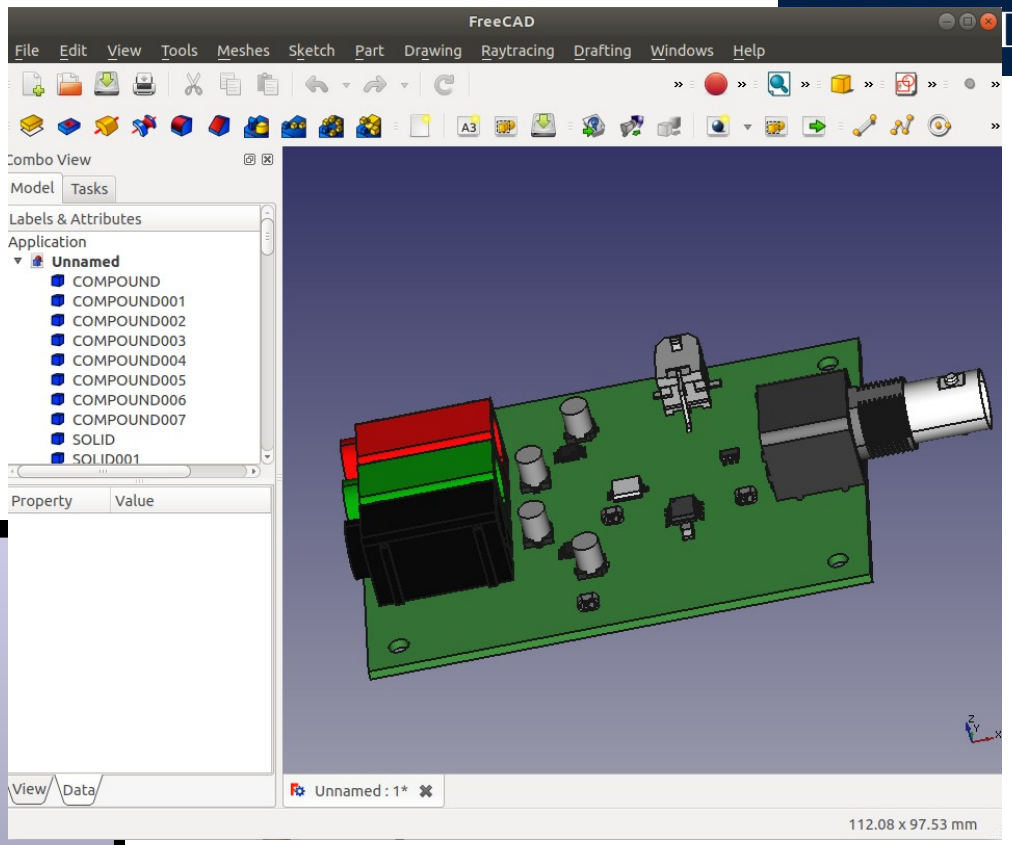


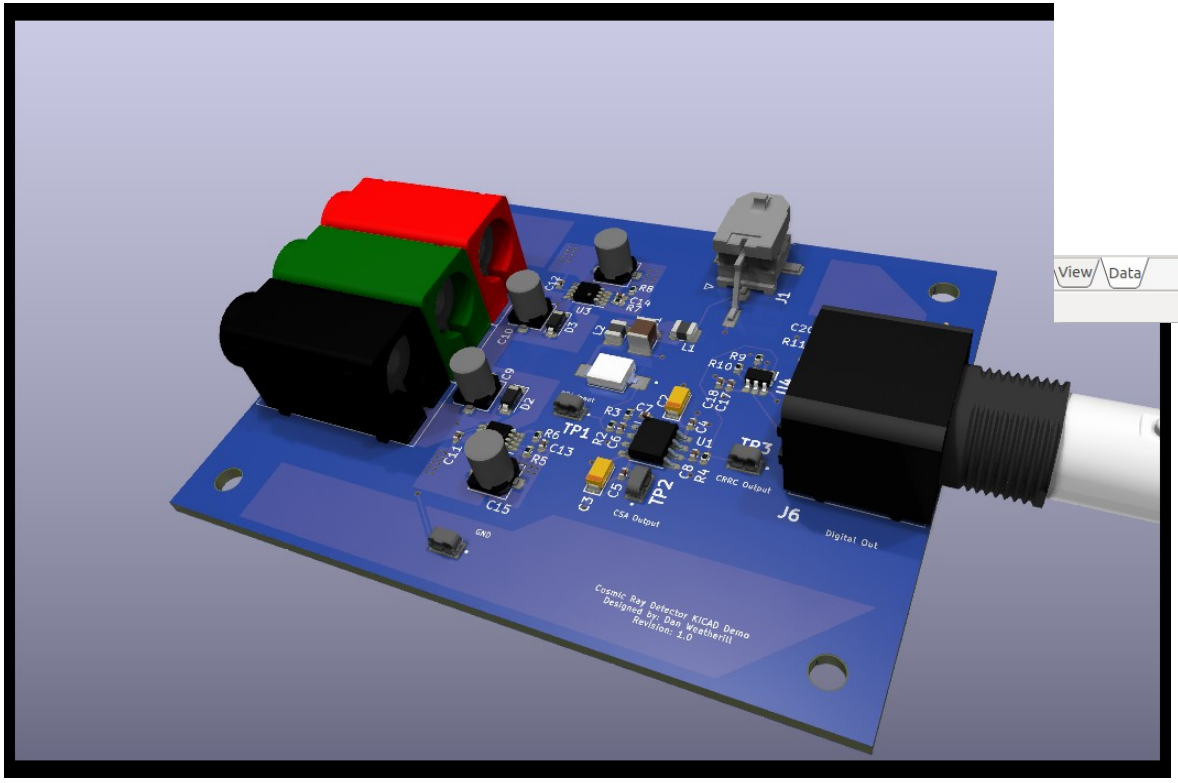
Gerber output Is found in the “plot” dialog. KiCad can export full Gerber X2 and includes an advanced viewer – which can even highlight nets and components (this info is included in Gerber X2 files, welcome to 2023!
ALWAYS check your gerbers layer by layer with a 2nd tool, be it another offline viewer (e.g. gerbv or CircuitCAM or similar), or an online viewer (https://gerblook.org). **MAKE SURE** you understand any unexpected discrepancies before sending them off!
(NB this includes misalignments, you may have got the coordinate origins wrong or similar!)

KiCAD – 3D MCAD Export

The 3D viewer is not just for show (though it can produce fairly pretty 3D renders as shown **below**)

It is useful to double check footprints, By downloading manufacturer's 3D models and making sure they fit





You can also export a .STEP file from kicad, which can then be opened by your mechanical engineer friends (**above**), though they probably have something better than FreeCAD

Walkthrough PCB Layout

We will show the basics of laying out a PCB from the project we looked at last time. Again, downloadable from the Indico in several forms:
1) “pcb_proj_blank.zip” – nothing done in the PCB except DRC setup and layer setup

2) “pcb_proj_noroute.zip” – the hardest part done, all the components placed, you get to do just the fun part, the routing!

3) “pcb_proj_someroute.zip” – some components placed, and some routing done, for you to fill in the rest

4) “pcb_proj_complete.zip” – completed PCB routed project, and included output Gerber/Excellion / IPC-D netlist / BOM / drawing / STEP model files for you to look at

Also available on github:

“https://github.com/weatherhead99/AITL_kicad_pcb_design” - there are tags and branches for the various stages of completion shown in the demos & the export files on Indico.

Finally NB: some schematic tweaks and a”magical” BOM export have appeared in these files since last time, so if you carry on from previous project (Tuesday) you may find minor differences. Sadly we didn’t have time to go over fully BOM export in these 2 hours)

PCB Layout Steps

UNIVERSITY OF OXFORD

Steps to make a PCB:

1) import components from schematic editor. Setup DRC rules, netclass rules & board stackup

2) manually draw board outline & place some mounting holes

3) place components (will show basics then skip over, this is ~90% of PCB design time)

4) route tracks (will demo the manual “PnS” router), tidy up silkscreen, mask & solder layers

5) Tidy up silkscreen, check mask, solder paste layers etc

6) make the DRC pass! (repeat steps 3,4,5 as needed)

6) export for manufacture (gerbers, netlist, PCB drawing & MCAD model)

7) review gerbers

Thanks!

UNIVERSITY OF OXFORD

Happy PCB Designing!

Questions, comments etc greatfully received:

Daniel.weatherill@physics.ox.ac.uk

Very happy to be contacted about “stupid questions” re: KiCad or any PCB designs. I’ve done somewhere around 32 over the years now (in KiCad, altium, Eagle, DipTrace-urrrgh) so do know a little bit about what I’m doing

The docs for kicad are now quite good: https://docs.kicad.org/6.0/en/

There are also pretty good free video tutorials online, and in addition a very good (but pricy ~£50) commercial eBook and course teaching you the details of the program available here: https://techexplorations.com/kicad/

This course also includes a lot of “basic” PCB design stuff so is probably not ideal for experienced designers coming from other programs