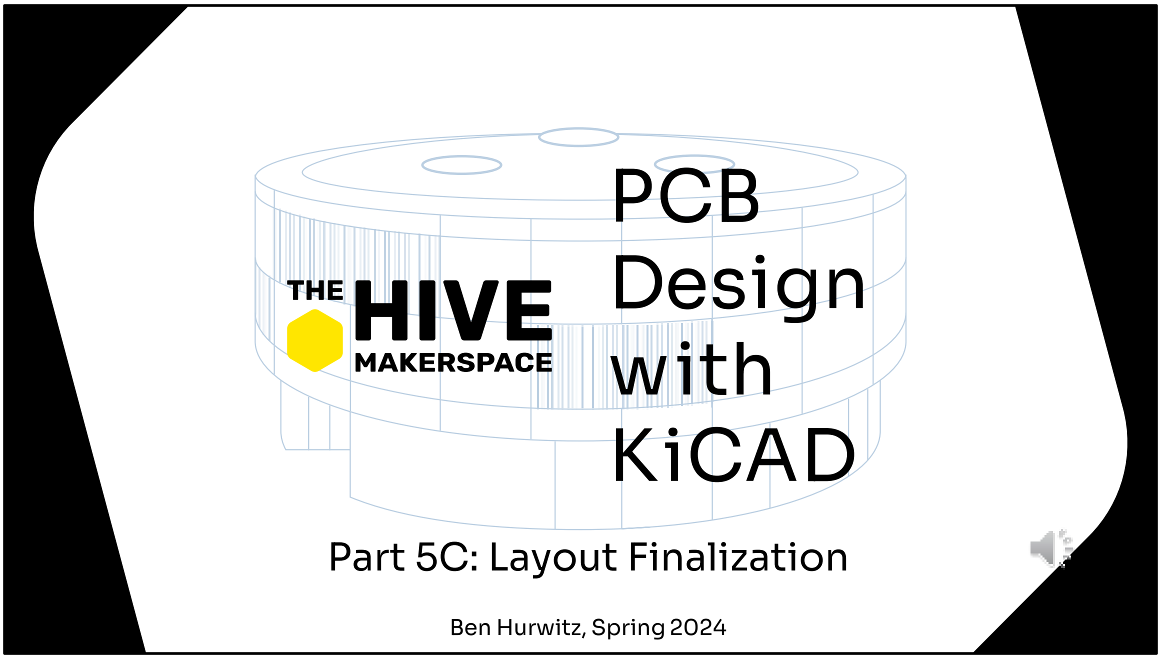


Hi all, welcome to The Hive's series on PCB Design with KiCAD. My name is Ben, and in this series, we've been walking through the PCB design process using KiCAD as our electronics design software.

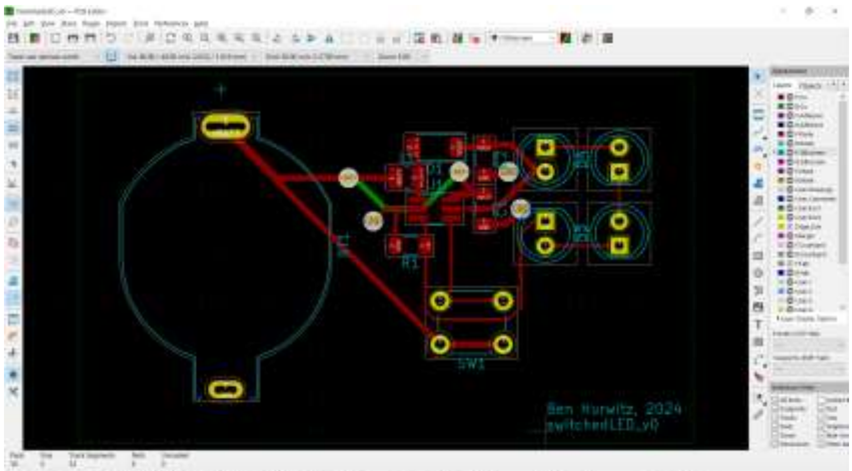
Part 5 has been focused on the layout portion of the design.

In the last video, part 5B, we placed all the components and routed them together into a single cohesive layout.

In part 5C, we'll finish up the design process by confirming the board visually and through DRC, and then plotting our gerber files for fabrication and assembly.

Let's get started.

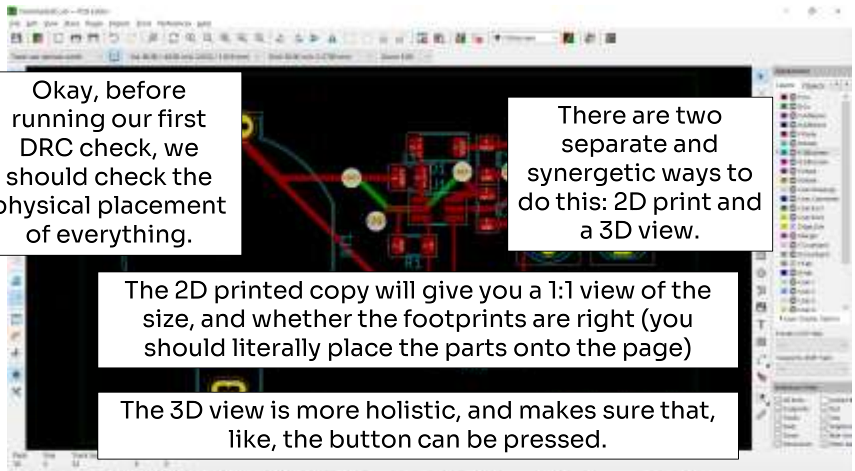
Layout



My final layout looks like this. Yours may well look different, and that's okay. As long as everything is connected and there are no more air wires, those thin white ratsnest lines, then we can continue.

If you still have air wires in your design, pause the video here and finish routing first.

Print a copy/3D view



Okay, before running our first DRC check, we should check the physical placement of everything.

There are two separate and synergetic ways to do this: 2D print and a 3D view.

The 2D printed copy will give you a 1:1 view of the size, and whether the footprints are right (you should literally place the parts onto the page)

The 3D view is more holistic, and makes sure that, like, the button can be pressed.

Print a copy/3D view

Printing a 2D copy is easy.

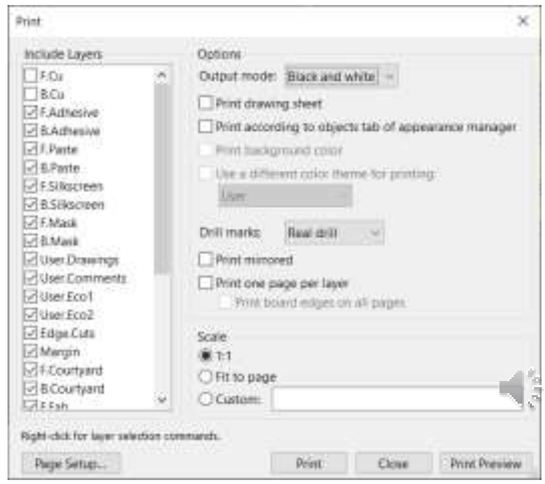
“File” > “Print”

Usually the default set of layers is good for mechanical scale.

Leave the drill marks as “Real drill” or the size will be wrong.

And keep the scale to 1:1.

Do a print preview first!



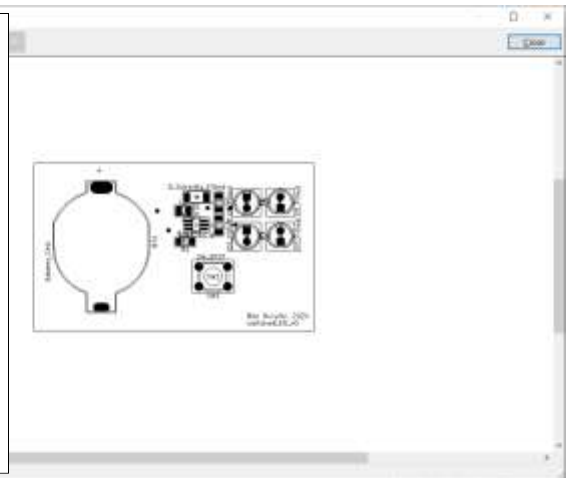
Print a copy/3D view

Here's the preview.

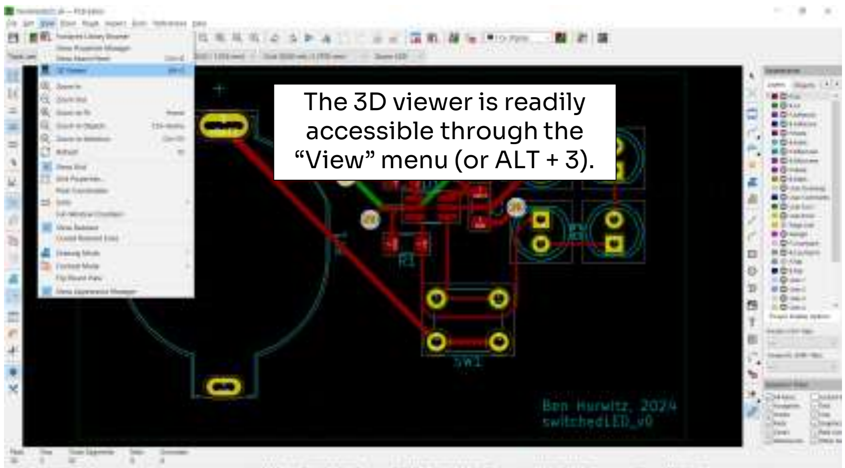
The routing isn't relevant because what we're looking at is the physical size and shape of the board and parts.

Physically place components on the board to confirm footprints.

Very helpful to avoid accidentally using the wrong footprint or a super tiny package.



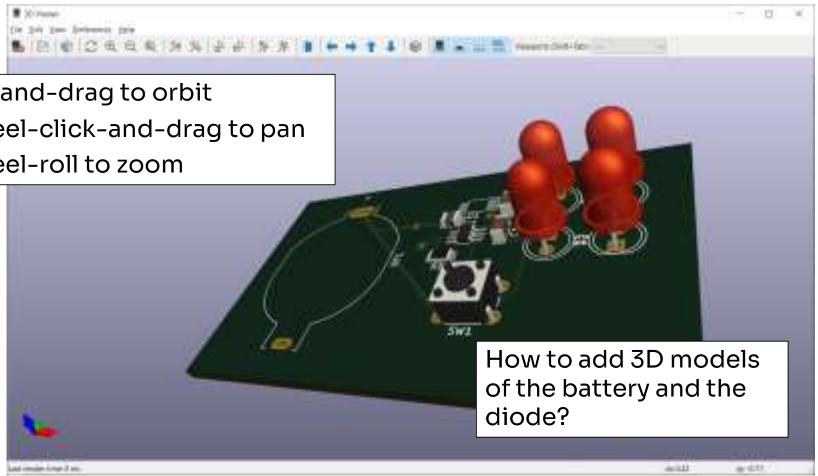
Print a copy/3D view



Print a copy/3D view

Print a copy/3D view

Left-click-and-drag to orbit
Scroll-wheel-click-and-drag to pan
Scroll-wheel-roll to zoom



How to add 3D models of the battery and the diode?

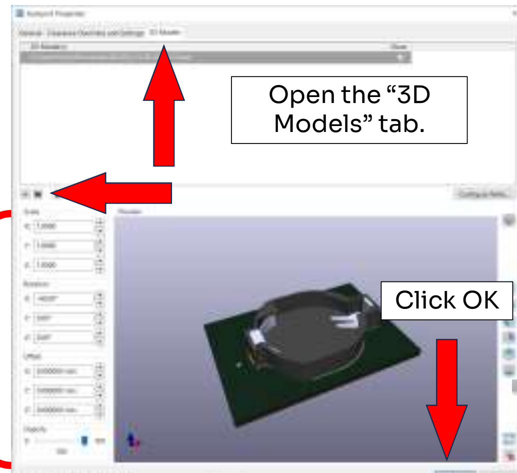
Print a copy/3D view

From the layout view, open the properties window of the footprint in question (right-click, “Properties”).

Open the “3D Models” tab.

Open and locate an acceptable file type (STEP or IGES, mostly)

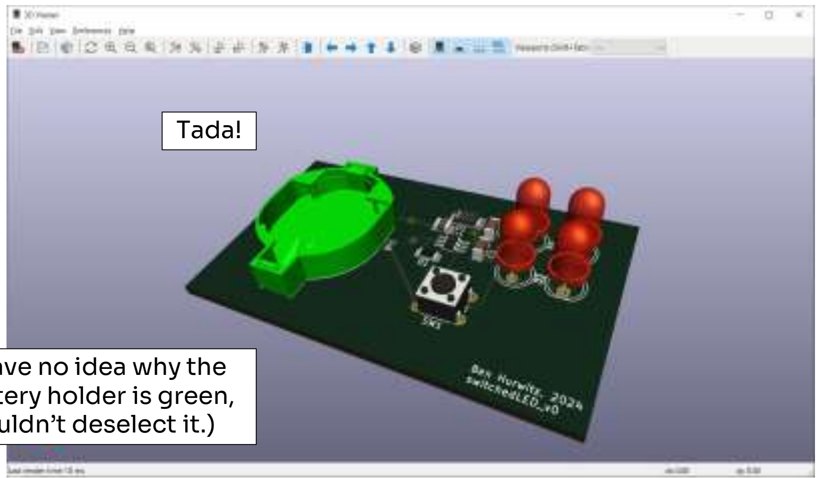
Adjust the position as necessary to sit flush on the board.



Click OK

A well-designed 3D CAD model should have the origin aligned properly such that one or two rotations should have the part sitting flush, if that.

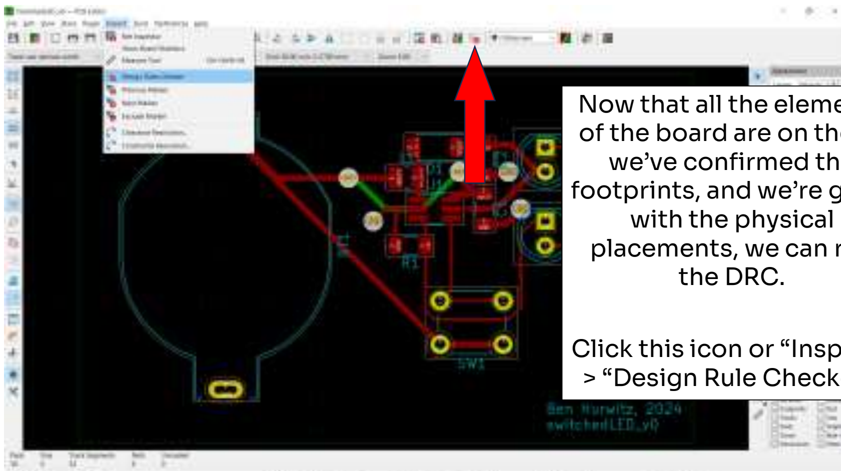
Print a copy/3D view



Tada!

(I have no idea why the battery holder is green, I couldn't deselect it.)

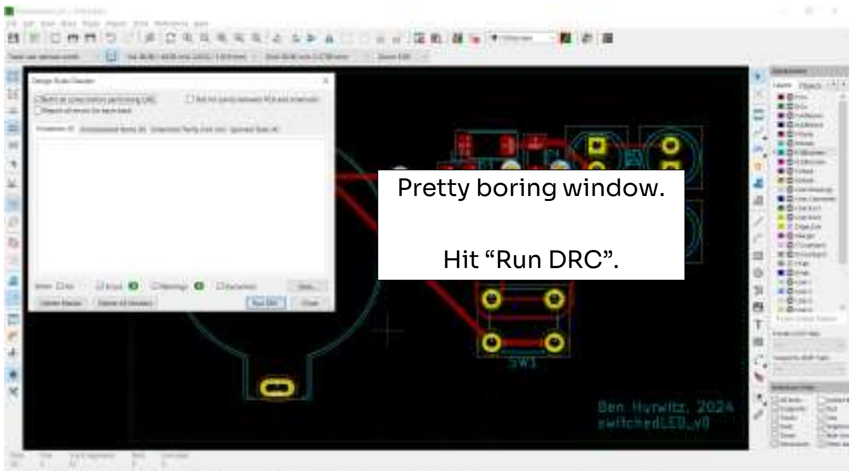
Layout



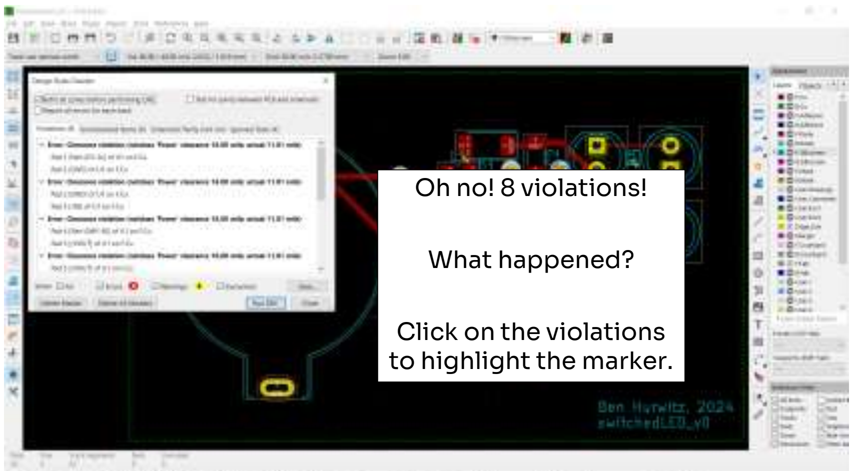
Now that all the elements of the board are on there, we've confirmed the footprints, and we're good with the physical placements, we can run the DRC.

Click this icon or "Inspect" > "Design Rule Checker"

Layout


Pretty boring window.
Hit “Run DRC”.
Ben Hurwitz, 2024
switchedLED_v0

Layout

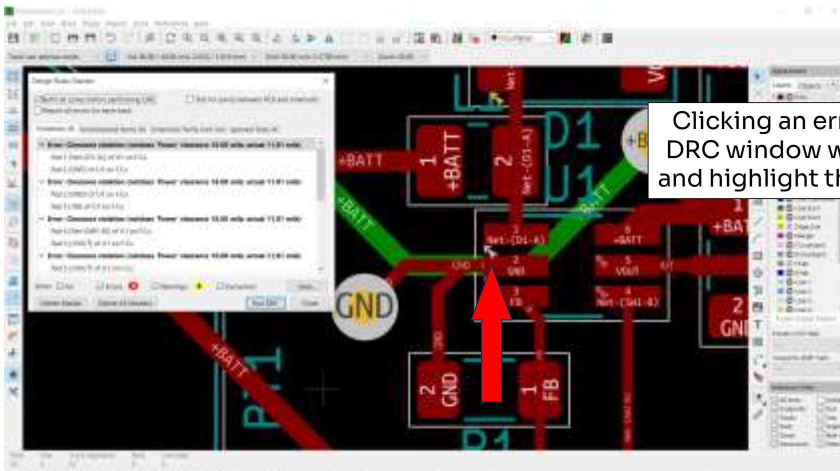


Oh no! 8 violations!

What happened?

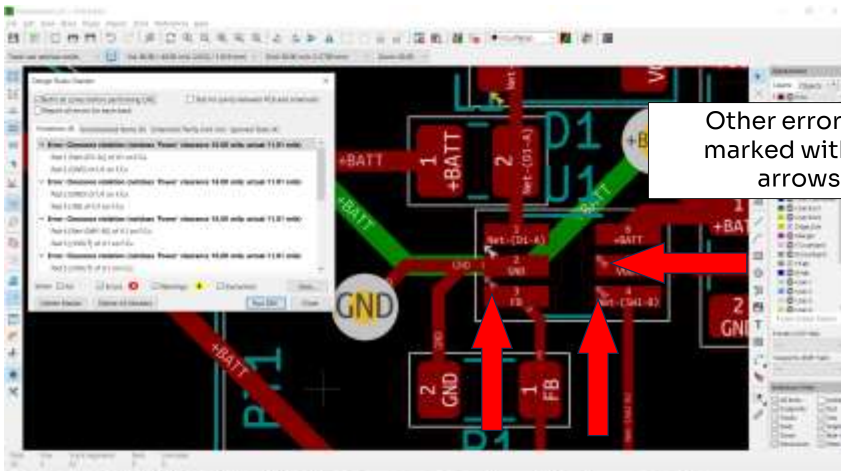
Click on the violations to highlight the marker.

Layout



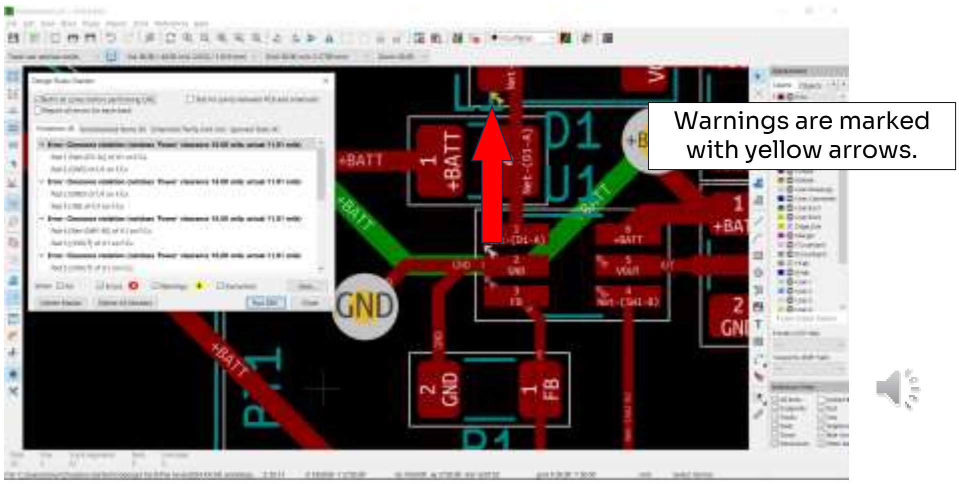
Clicking an error in the DRC window will center and highlight the marker.

Layout

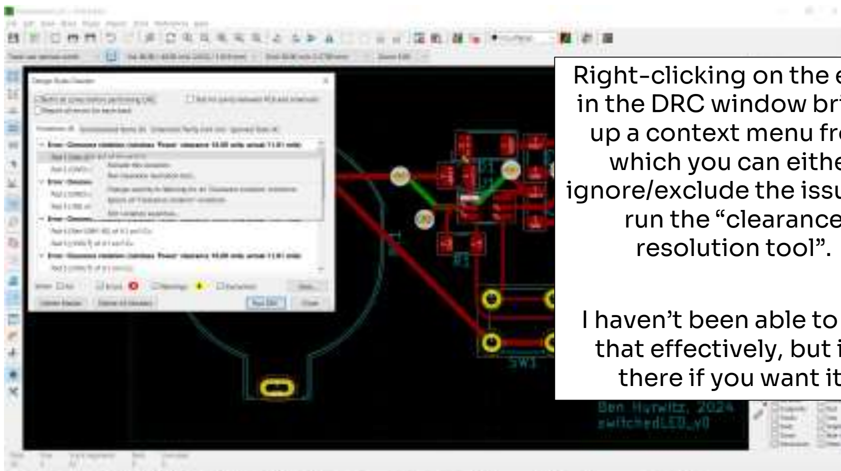


Other errors are marked with red arrows.

Layout



Layout



Right-clicking on the error in the DRC window brings up a context menu from which you can either ignore/exclude the issue, or run the “clearance resolution tool”.

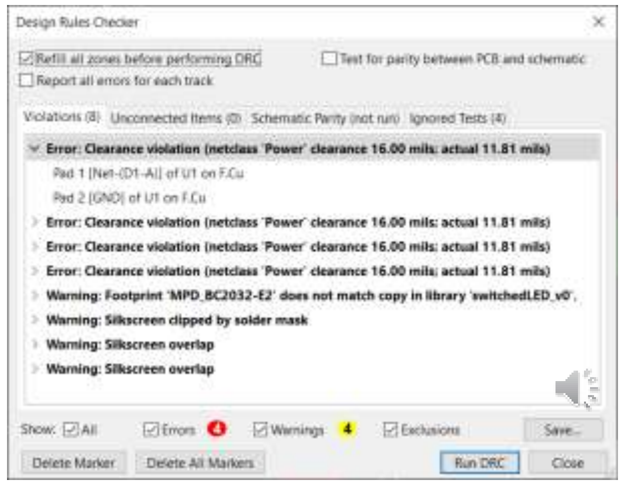
I haven’t been able to use that effectively, but it’s there if you want it.

Layout

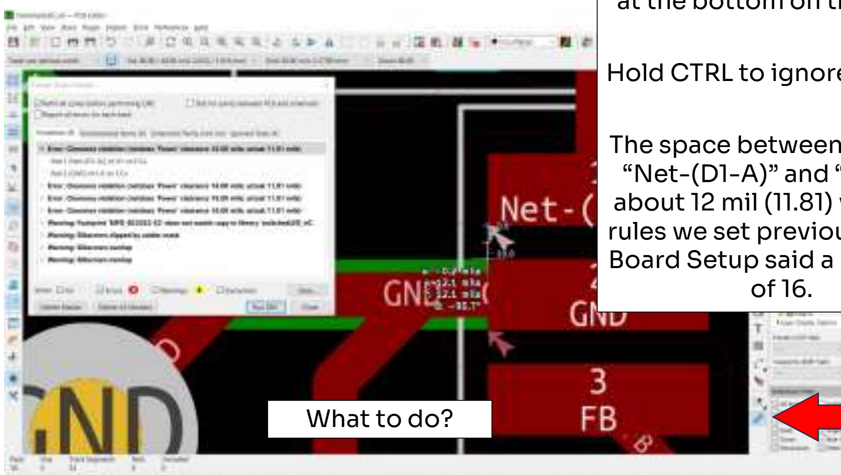
What are the errors?

The first four are “clearance violations” with things on the “Power” netclass, i.e. the clearance minimum was broken.

This means that something on a net within that netclass is too close to something not in that netclass.



Layout



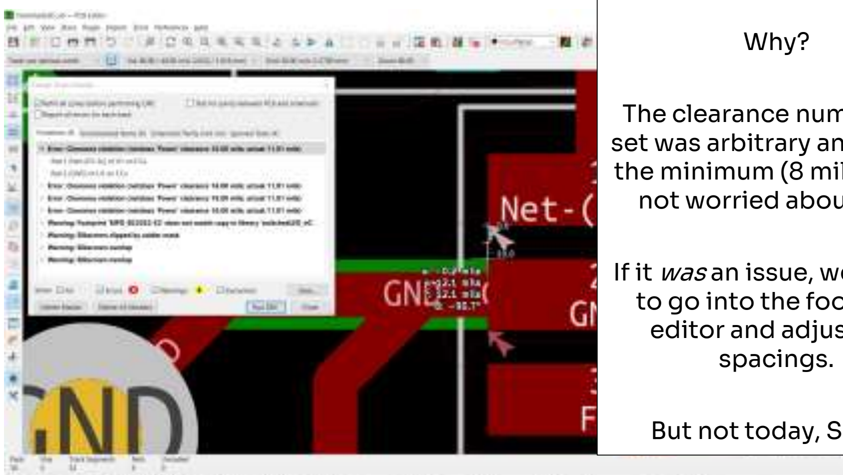
What to do?

We see the issue by using the measure tool (CTRL+Shift+M, or the icon at the bottom on the right).

Hold CTRL to ignore the grid.

The space between the pads “Net-(D1-A)” and “GND” is about 12 mil (11.81) when the rules we set previously in the Board Setup said a minimum of 16.

Layout



Nothing – we can safely ignore this error.

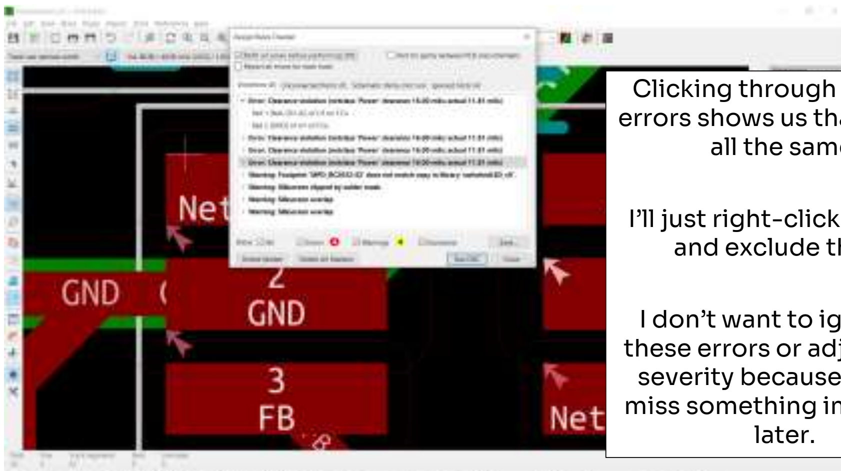
Why?

The clearance number we set was arbitrary and above the minimum (8 mil), so I'm not worried about this.

If it *was* an issue, we'd have to go into the footprint editor and adjust the spacings.

But not today, Satan.

Layout



Clicking through the four errors shows us that they’re all the same.

I’ll just right-click them all and exclude them.

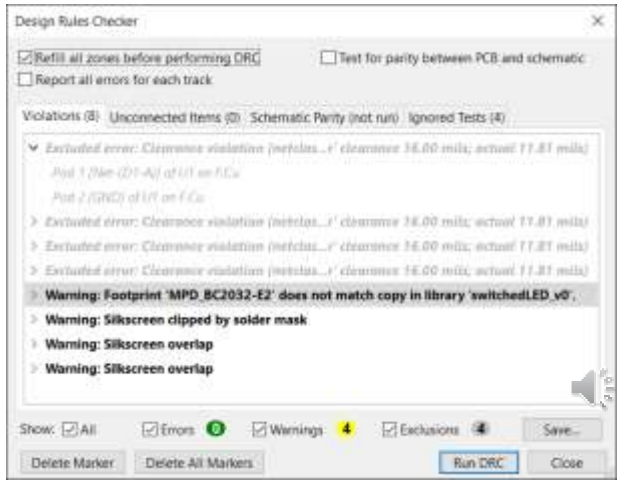
I don’t want to ignore all these errors or adjust their severity because I might miss something important later.

Layout

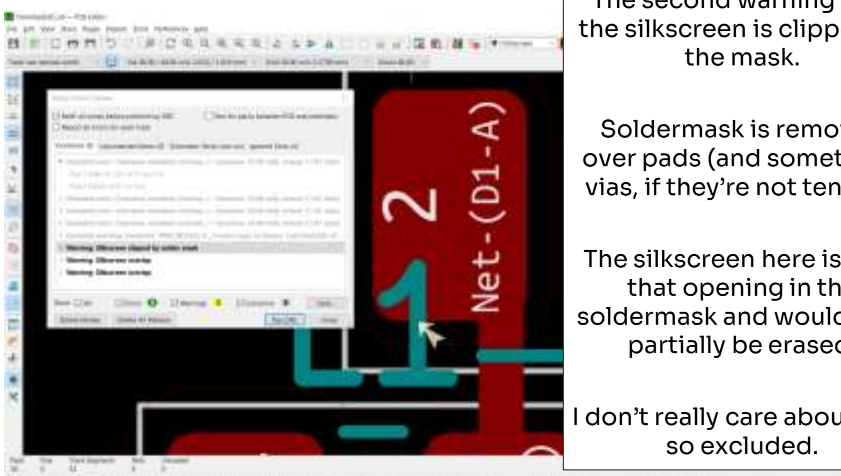
The first warning says that the battery footprint doesn't match what's in our library.

Well, of course it doesn't. We edited it.

Excluded.



Layout



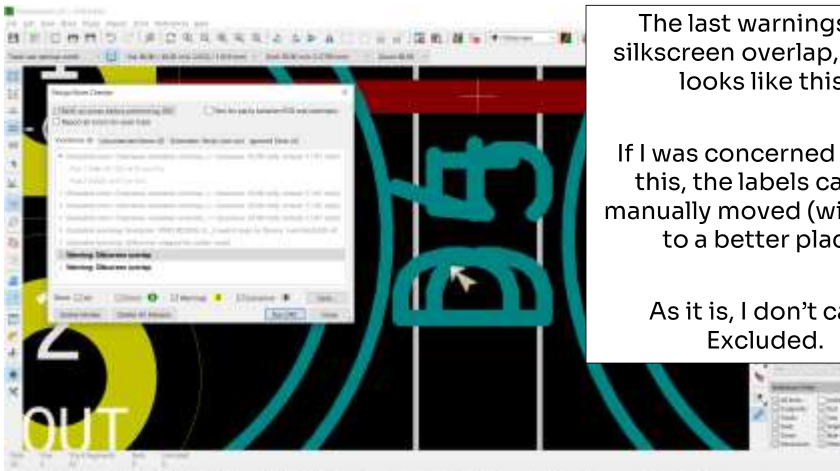
The second warning says the silkscreen is clipped by the mask.

Soldermask is removed over pads (and sometimes vias, if they're not tented).

The silkscreen here is over that opening in the soldermask and would thus partially be erased.

I don't really care about this, so excluded.

Layout

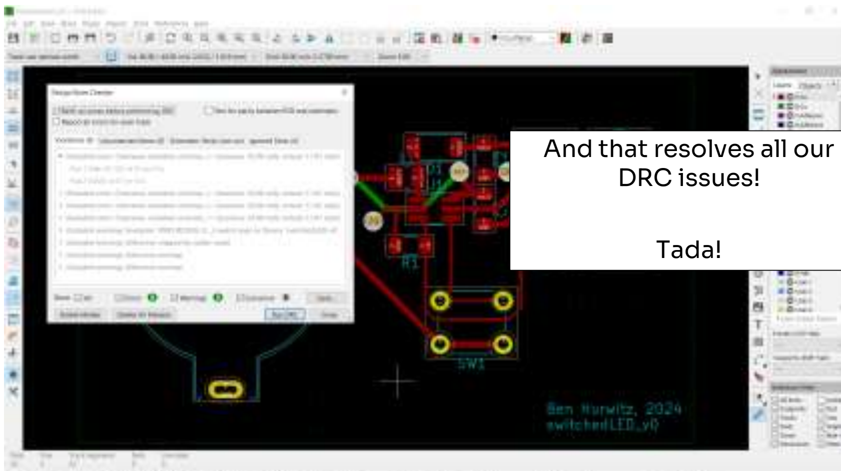


The last warnings are silkscreen overlap, which looks like this.

If I was concerned about this, the labels can be manually moved (with “M”) to a better place.

As it is, I don’t care. Excluded.

Layout



And that resolves all our DRC issues!

Tada!

The design is done!

Congratulations!

We have a fully designed
and checked board.

One final thing to do – plot the gerber files

Gerbers

Gerber files are special text files that define the polygons on each layer of your design.

You typically need to sent them to the fab house for fabrication.

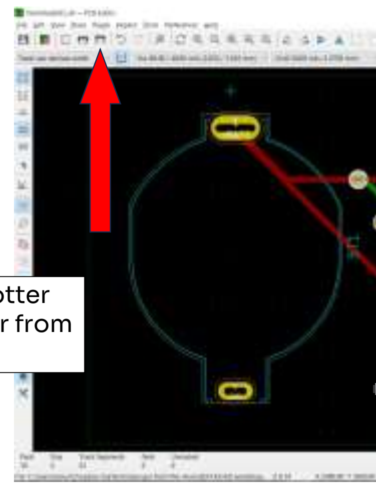
Some take KiCAD projects or board files directly these days.

Read the fab house's instructions for plotting these carefully!

It's easy to do something wrong and get a bad board.

Gerbers

Open the gerber plotter either with this icon or from “File” > “Plot”.



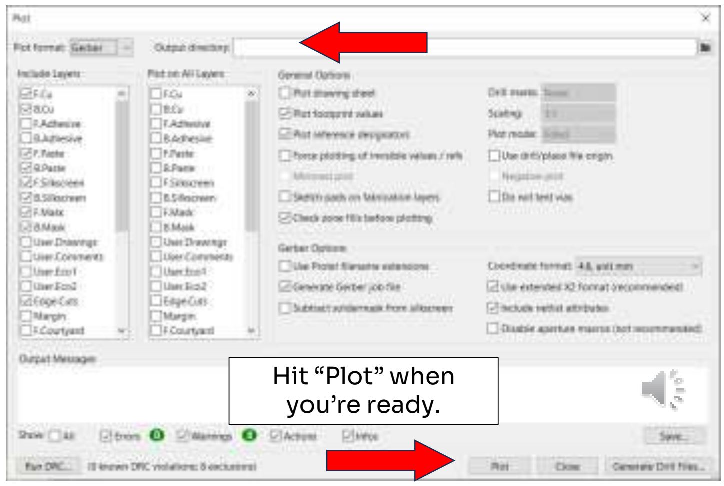
Layout

This is the plotting window.

Set an output directory for where to save the gerber files to.

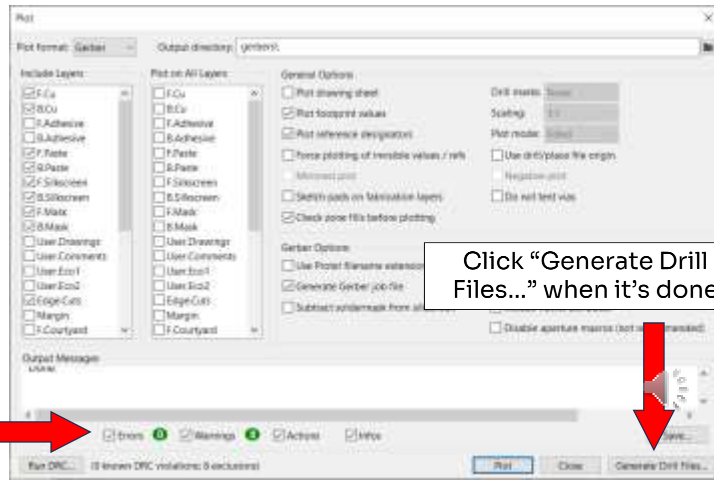
Read through your fab house's instructions carefully.

They will have details of what needs to be set in the middle.



Hit "Plot" when you're ready.

Layout



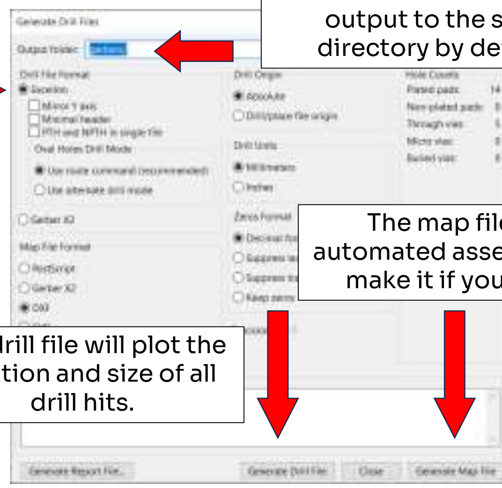
Click “Generate Drill Files…” when it’s done

Hopefully there are no errors.

Layout

Excellon is usually okay, but again, read the fab house instructions.

The rest of this will also be determined by your fab house.



The drill file or files will output to the same directory by default.

The map file is for automated assembly. Only make it if you need it.

The drill file will plot the position and size of all drill hits.

Gerber Viewer

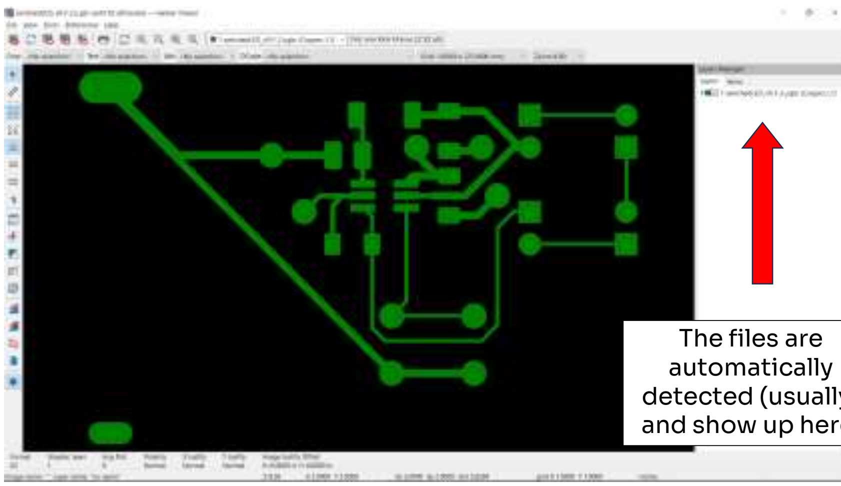
Gerbers can be viewer using the Gerber Viewer

Gerber Viewer

Gerbers can be opened with this icon

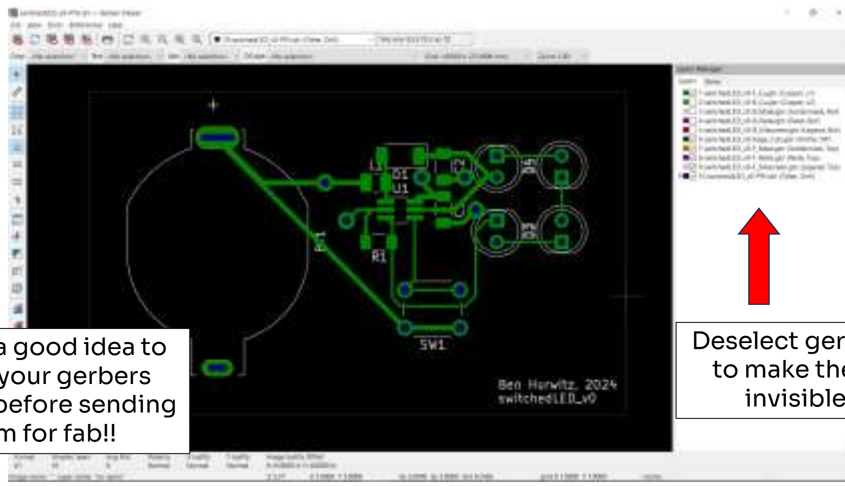
And excellon drills with this one.

Gerber Viewer



The files are automatically detected (usually) and show up here.

Gerber Viewer



Always a good idea to check your gerbers directly before sending them for fab!!

Deselect gerbers to make them invisible

You made it!

Congratulations! You have a fully designed board that is ready to be sent for fabrication!

At this point, you could stop the video, but the net few slides I'll provide you with some additional rescourse and information for you to keep in your back pocket (or as a bookmark, if people still use those).

Next steps

- If you're interested in actually making this PCB:
  - You can fabricate the board here at The Hive yourself, or you can send the gerber files off to have it done for you
  - Email the Hive PCB group (hive-pcb@ece.gatech.edu) for the BOM, and we'll see if the Hive can purchase the components.
- Future revisions:
  - Consider re-selecting some parts for SMD only design, and then redo the layout using the backside for parts too.
  - The LEDs could be on a separate board that plugs in for a more point-and-shoot flashlight

Further Resources - KiCAD

- KiCAD documentation (https://docs.kicad.org/)
  - Everything about KiCAD and more, including "Getting Started"
- KiCAD library conventions (https://klc.kicad.org/)
  - The "Symbol" and "Footprint" subsections can help demystify the naming standards
- KiCAD forums (https://forum.kicad.info/) and resources
- KiCAD DRC templates (unofficial – GitHub)
- Plugin and Content Manager
  - Some are made by fab houses for official integration
  - Plenty of useful add-ons to the software

Further Resources - Design

- Design guidelines by David L. Jones (PDF) <- The real OG.
- Via guidelines (Cadence) and trace width requirements (Advanced Circuits)
- Phil's Lab (YouTube)
- SnapMagic has a desktop app with KiCAD integration
  - I just learned about this, but might be useful?
- Random, but The Hive's standard resistors are 7mm long with a body diameter of 2.5mm and a 0.6mm diameter leads.
  - For when you use those and need to select one of the many footprint options.

Further Resources – Fab

- Part sourcing: Digikey, Mouser, Octopart, Sparkfun, Adafruit, DFRobot, Amazon (YMMV), eBay (rare/hard to find)
- Fabrication houses: PCBShopper (price comparison), The Hive (local), OshPark (US, low cost), Advanced Circuits (US, mid-range), Epec (US, very fancy), Sunstone (US, mid-range), JLC PCB (CN), All PCB (CN)
  - Many have EDA-specific instructions

The Hive doesn't officially endorse any of these suppliers or fab houses; they all have plusses and minuses.

Further Resources – Misc

- Google.
  - Seriously, this is the thing you should get proficient with.
- Guides on design, KiCAD, and other EDA software are available on YouTube, Adafruit, Sparkfun, and many (many) more.
- Feel free to stop by The Hive to ask questions, as well. We're here to help with your design and fab!

And with that, we’re done with part 5C and with the design! Congratulations. A PDF of this video is available as well, linked in the description and hosted on The Hive’s Wiki.

The remaining four videos cover library management and model creation. It’s less exciting, but if you’re thinking about doing more design, it’s really valuable to understand how to keep your parts organized and ordered for later use and reuse. Part 6, which is next will look at symbol libraries, with some duplicate material from part 4. Part 7, which is split into three videos, will cover footprint libraries, and custom footprint generation.

Hope to see you there!