Kicad Primer

Contents

1 Main List

KiCad PCBNew Primer: A quick recap, to refresh my KiCad PCB abilities[1]

Summary: This document should be read before starting on a PCB layout with KiCad. These tips may apply to other PCB software as well.

- Route Power First: Route power first, and pay attention to how many layers you will need. Depending on how many signals, and where they are going, you may need a 4 or higher layer board. 144LQFP was difficult using 100% of the pins, but possible on a 2 layer board.

- Case and Mounting Holes: Put the mounting holes into the PCB early, to avoid rerouting layer. Same with the case.

- Keyboard Shortcuts: Use keyboard shortcuts. X to choose 'add trace', PageUp and PageDown to cycle through layers, there is a hotkey for changing layer types. O will add a component (enter or left click after selecting O)

---

[1]Best viewed on actual paper not a computer screen.

- Measuring: There is a ruler in KiCad. Extremely useful.

- Ground Pours: If you want to do a ground pour that is split, there is the keepout area. This keepout area has an option for component or fill keepout. Choose the fill keepout. Test on a blank project to see workflow.

- Via Size, Trace Size, Fab House Guidelines: Set the fab house guidelines into KiCad before starting, otherwise you will have to change items later. e.g. if you set the Vias to be too small, you will have to edit all vias later. There are templates available with DRC guidelines for OSHPark, as an example.

- White Background: In 5.0.0rc2 and newer (assumed) change the background colour to white by clicking on the option. It may be greyed out, but still selects.

- Board Edge: Edge.Cuts and use a 1 mil or .001 inch add graphic lines on the edge cut layer.

- Resources: There are a variety of resources available, [**?**, **?**] on PCB design. Not including textbooks, and forums.

- Modular: Make separate boards, and reuse what you can. Some things are trivial to layout, and don't need separate boards. Anything that is difficult to layout, consider using it's own board.

- Net names: Give net names to tracks. This will make identification easier on layout. E.g. see here how I've added labels to nets that might not need them.

- Component Values: If you are building by hand, add component values to the footprints silk screen to make it easier to identify. See picture.

2   Miscellaneous Tips

2.1   Silk Screen

Silk screen that goes over the board edge, can cause graphical glitches. Oshpark uses software to ignore anything over the board edge, but other pcb fabs might not.
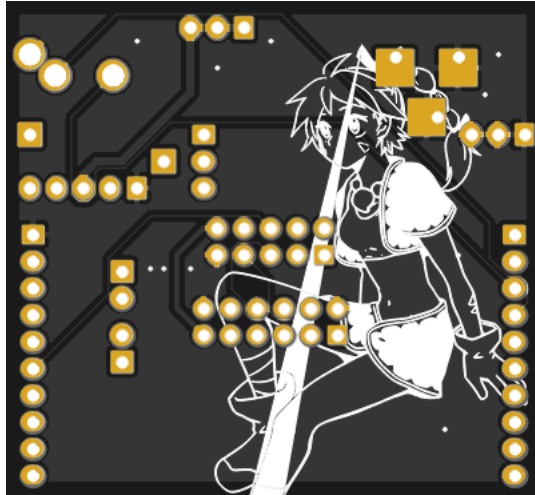
Figure 1: Just one possible gerber error. Here, from Silk Screen being over the board edge.
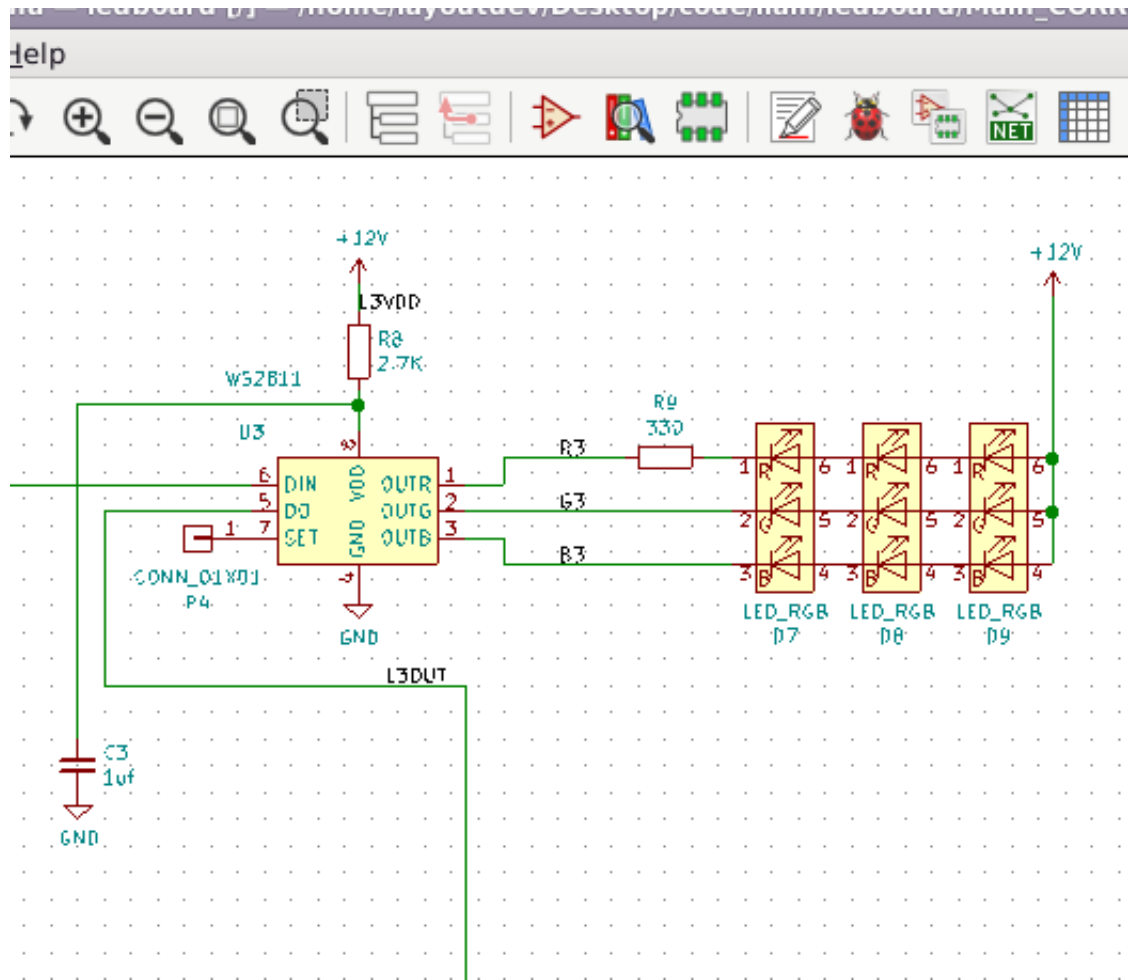
2.2 Net Names

Figure 2: Giving Net Names to nets that may not need them can help in layout later. See these images...
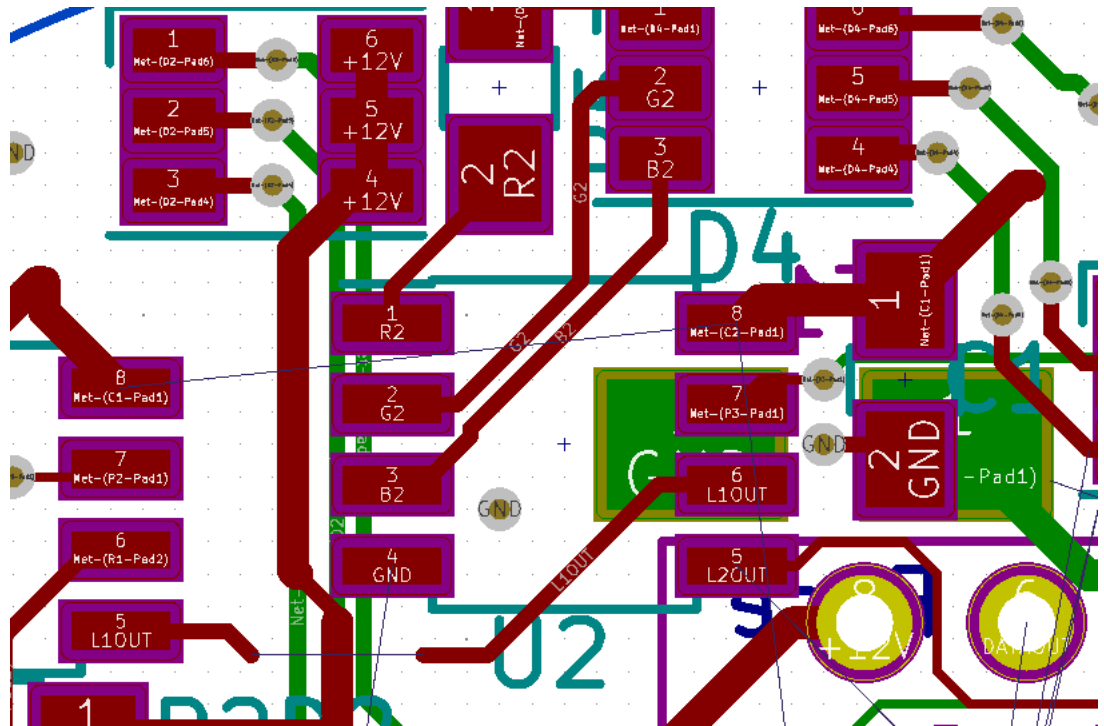
Figure 3: This converts the mystifying names of NET-(P2-Pad4) to something human readable like R2, G2, B2.
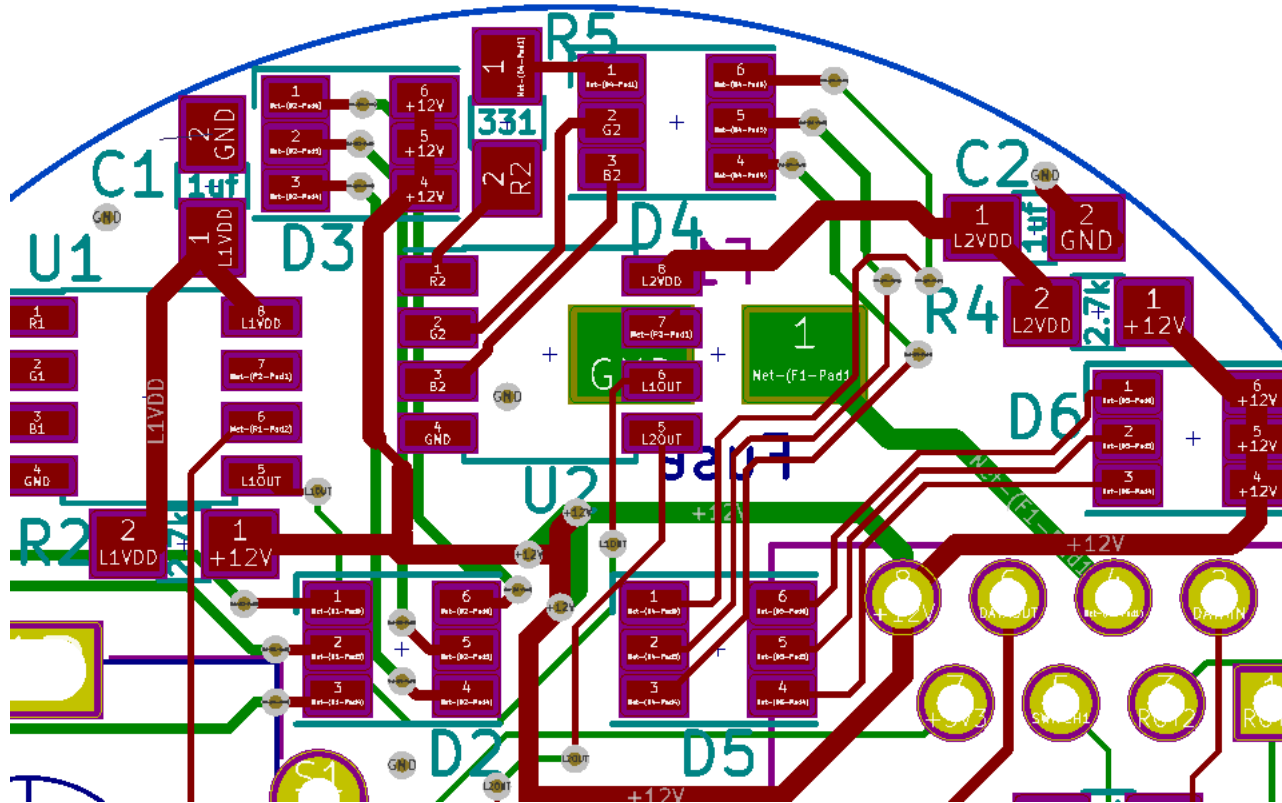
Figure 4: By adding a text of 0.5 size to 0805 footprints, you can easily fit within the provided kicad footprint. This makes assembly easier. See here, labels such as 1uf, or 2.7k.

2.3 How to import graphical pictures into Kicad

This is how I convert pictures to silkscreen. First, download any picture that can be converted to Black and White. Monochrome pictures are an option, or you can take a coloured image and convert it in GIMP. To convert a coloured image in GIMP do the following:

- Open picture in GIMP

- Set **image - mode** to **Greyscale** (top menu)

- Now we get a preview, but this is varying levels of Black and white,

we want only two colours, so choose **colours - threshold** (top menu) and choose the amount you want (the defaults are usually acceptable)

- Use the **Select By Color Tool** from the tools menu and choose the black that will be turned into white for the silkscreen.

- Copy that (top menu - edit - copy)

- Paste it as a new layer (top menu - edit - paste as new layer)

- Hide the old layer.

- Select all the black on the new layer with the color tool

- **edit - fill with bg colour**, if the bg colour is white.

- Resize if needed, and export as png

A short synopsis of what this process entails for those more intimate with photo editing programs is:

Get a black and white picture. Make the part you want white. Make the background transparent. You should now have only a white ink layer of what the silk screen will be, with a transparent background. Save as png. Import into bitmap 2 component.

References

[1] Spark Gap Podcast: 21,22, and 48, http://thesparkgap.net/the-spark-gap-podcast-episode-21/, http://thesparkgap.net/the-spark-gap-podcast-episode-22/, http://thesparkgap.net/the-spark-gap-podcast-episode-48/

[2] David Jones PCBDesignTutorialRevA, http://alternatezone.com/electronics/pcbdesign.htm, http://alternatezone.com/electronics/files/PCBDesignTutorialRevA.pdf