

Basic PCB Design using KiCad

USU SPAC spring 2020

Nikolas Clark

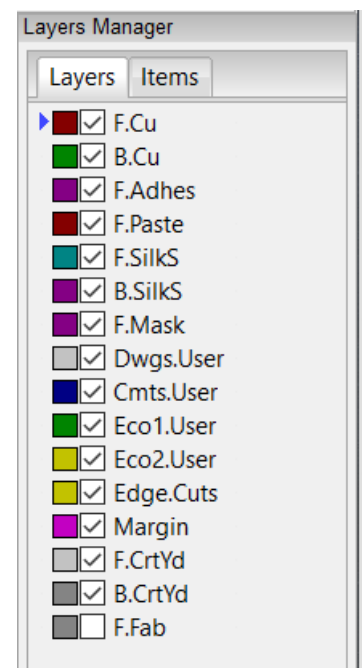
Lecture Notes

Most of the lecture outline won't be covered in these notes. Some of the details in lecture are best explained with simple diagrams and are included here. I also wanted to list some helpful links that can help with further research and design.

PCB Layers

KiCad prefixes the various layers with F and B for front and back of the board. Inner layers will be prefixed with In1, In2, and so on as necessary for boards with more than 2 layers.

- Cu – Copper. All pads and holes for components and all traces connecting them must be on a copper layer. Two sided boards are very common, but are more difficult to produce. To produce a two sided board requires two different solders with different melting points. If you have to build a board yourself try and stick to putting components on just the front copper layer.
- Paste – Solder Paste. This layer is used to create a stencil for your design.
- SilkS – The silkscreen is a layer that allows for text and drawings to be added to the board. This is used to label parts which is very important for manufacturing (especially if you're not the one doing it). It is also important to label which pin/pad is "1" to ensure that parts are soldered onto the board with the correct orientation.
- Mask – Solder Mask. This layers defines where the manufacturer should place solder mask onto the board. Mark in KiCad where you do not want soldermask to be applied. Soldermask is what makes a



PCB traditionally green. It protects all of the layers beneath it from moisture and dust, and controls the flow of molten solder. Also called solder resist.

- Edge.Cuts – This layer defines the outline of the board. In KiCad this can be used to define cutouts inside the board as well that will not be plated (this can be used to create unplated mounting holes for instance). If you need a plated hole in your design (to create a chassis ground for instance) use a footprint instead.
- CrtYd – Courtyard. Components often require additional space beyond the size of the pads. The courtyard layers shows the physical size of the components.

Traces and Vias

Traces are a PCBs version of wires. We use them to connect components together. When drawing a trace we must choose the width of the trace. In general thicker traces can handle higher amounts of current. KiCad has a trace width calculator that can be helpful for this decision. Manufacturers have a limit to how thin of a trace they can reliably produce. Oshpark's is 6 mils.

Best/Common practice is to avoid 90 degree angles in your traces. In the end this often simplifies not complicates a design as it allows you to make more direct connections between parts.

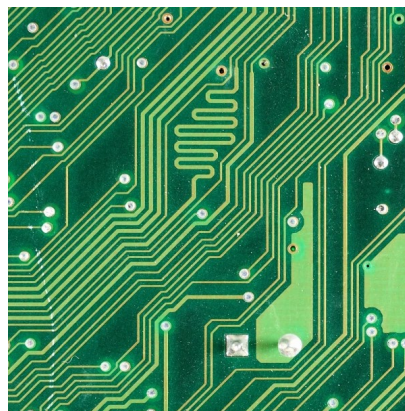


Figure 1: Traces on a PCB. The silver circles are vias.

Vias are plated holes drilled into a PCB to connect layers together. While there are several types of vias, Oshpark is only able to manufacture the most basic and common type, through-hole. This via connects all layers on a board. This means that traces will often have to avoid the via on other layers.

The other types of vias are shown in the figure below and require a much more involved manufacturing process.

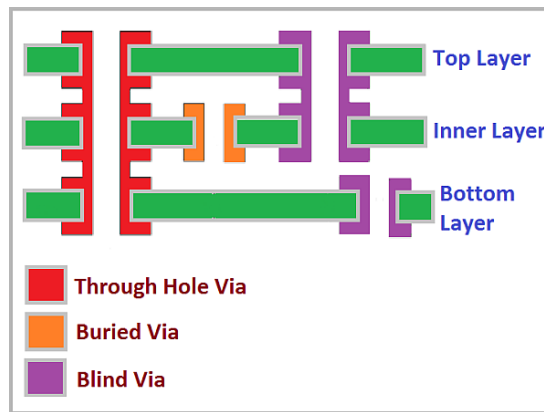


Figure 2: Types of vias

Via-in-pad is another relevant technique. This involves placing a via directly under the pad of a component. I have been advised to avoid this in amateur designs as it often makes it difficult solder as the solder is drawn into the hole instead of attaching correctly to the component. This technique is however employed successfully in many professional designs using techniques detailed here:

<https://electronics.stackexchange.com/questions/39287/vias-directly-on-smd-pads>

Design Tips

Here are some design recommendations I have found researching online. I make no promises that all of the information here is correct however, I only know that they have worked so far in all of my designs.

1. Identify what each part of your circuit does, and divide the circuit into sections according to function.
2. Long traces can pick up electromagnetic radiation from other sources
3. The different sections of your circuit should be arranged so the path of electrical current is as linear as possible.
4. Each section of the circuit should be supplied power with separate traces of equal length. This is called a star configuration, and it ensures that each section gets an equal supply voltage. If sections are connected in a daisy-chain configuration, the current drawn from sections closer to the supply will create a voltage drop and result in lower voltages at sections further from the supply.
5. Some double layer PCBs have a ground layer, where the entire bottom layer is covered with a copper plane connected to ground. The positive traces are routed on top and connections to ground are made with through holes or vias. Ground layers are good for circuits that are prone to interference, because the large area of copper acts as a shield against electromagnetic fields. They also help dissipate the heat generated by the components. (All multi-layer designs that I

have seen include MANY ground planes. Often I have seen them placed in between every signal layer to help isolate signals and prevent interference).

6. Use thermal relief ties if connecting a pad to a large plane of copper. These make soldering easier as you thermally isolate the pad from the plane. Without thermal ties, you'll find that the heat from your soldering iron will be quickly dissipated through the ground plane. In KiCad thermal ties will look like the image on the right, not the left. The image on the left shows the default flooded via.



7. When using thermal ties you might be concerned about the trace width. My research says that the trace width calculator in KiCad uses formulas based on long traces without heat sinking. Thermal ties are very short and well heat sunk and thus do not fit that formula well.
8. To decide how much temperature rise your board can handle based on the operating environment and the type of PWB material used. Ten degrees is a very safe number to use for just about any application.
9. Consider the length of traces versus the frequency of the signal they will be carrying. The higher the frequency the more the traces need to be considered transmission lines and not ideal connections. If $\frac{\text{trace length}}{\text{wavelength}}$ is very small the effects of a transmission line may be ignored. If that ratio is greater than 0.01, it may be necessary to account for the phase shift due to the time delay, and for reflected signals. [Ulaby, F. T. *Fundamentals of Applied Electromagnetics*.]
10. For reference common communication protocols use the following frequencies:
 - I2C uses from standard to high-speed: (100 kHz, 400 kHz, 1 MHz, and 3.4 MHz)
 - SPI, UART depend on the configuration of the devices involved but can be in the Mhz range.

Useful Links:

Kicad Software: <https://kicad-pcb.org/>

KiCad Tutorials: <https://kicad-pcb.org/help/tutorials/>

Oshpark: <https://oshpark.com/>

Oshpark design rules for KiCad: <https://docs.oshpark.com/design-tools/kicad/kicad-design-rules/>

OshStencil: <https://www.oshstencils.com/#>