Recitation: Planning your layout

Trace: essentially a wire on your board
Plane: Big section of copper connected to one net

Step 1: Define board stackup

We'll be doing 2 layer boards, so there's not too much thought that needs to happen here. Both layers will be signal layers (they have traces on them as opposed to planes)

For anything more complex, you will quickly move to 4 layer (they are not much more expensive) A typical stackup is Signal, GND, PWR, Signal

For higher layer counts, you might do something similar, but now you have additional signal layers.

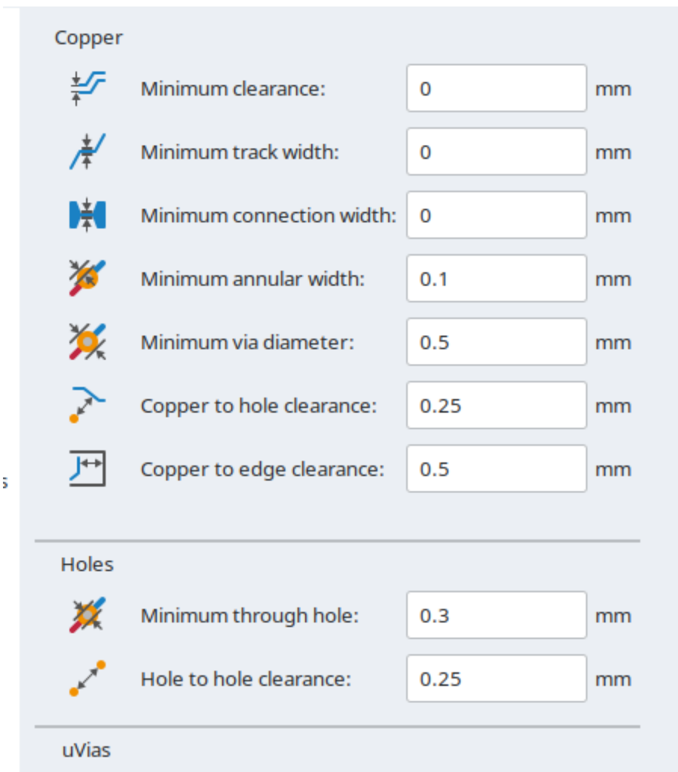
For some applications (e.x. digital boards with a lot of connections) you might designate signal layers for either horizontal or vertical runs. This helps traces not have to jump over each other so much, and it can also reduce crosstalk (since traces are orthogonal)

Step 2: Define design rules

Design rules keep us from doing things that can't be manufactured. They can be pretty complex, but most manufacturers should have a decent guide of what they can support (and at what price they can support it)

Step 3: Define board outline

At this point you probably want to define the shape of the board that you'll make. If you aren't particularly constrained by anything (such as the mechanical enclosure), you can try to estimate the amount of space you'll based on the number and size of components you're placing on the board. Keep in mind that connectors, buttons, and other human interfacting things frequently want to go on the edge of the board.



Some fairly standard design rules in KiCad

Step 4: roughly layout the rooms

- KiCAD does not have a formal concept of rooms, but you can still follow the same process. A room is a grouping of all of the components on one schematic sheet. Since these typically are primarily connected to each other, it makes sense to group them into a room and think of routing them separately.

- While doing this, consider a few things (which mainly boil down to "what is most important"):
    - High speed/frequency traces (such as USB, signal from FM antenna)
        - Typically want to keep these traces short

    - Noisy traces and areas (power converters especially, but also high speed traces)
        - Keep these away from noise sensitive areas
        - Consider shielding

    - Noise sensitive areas (mainly analog things like antenna connection, audio lines)
        - Keep away from noisy things
        - Consider shielding

    - Mechanical constraints
        - Consider where connectors need to go (probably edge of board)

    - Connections between rooms
        - Routing will be easier if connections that go between rooms are short

    - High voltage/current traces
        - High current needs large trace widths to provide low resistance
        - High voltage needs clearance so that racking/arcing/other bad things don't happen
        - Both of these mean that they will take up space
        - It is also typically better to route these on outer layers (for thermal reasons)

Step 5: Component placement

- Now start actually arranging the components within their rooms. Your components should be fully placed before starting any routing. It's a good idea to reference the schematic while doing this.

- Consider similar things to step 4.
- Try to get bypass capacitors as close to power pins as possible.

- Some chips (especially power stuff) will have recommended layouts that can be useful

Step 6: routing

Once everything is placed, we can draw our traces. Start with the most important traces in (or between) a room. These are the traces that we were giving special consideration earlier

As you start doing this for the first couple times, you will likely gain insight about how you should have placed things better. It's ok and recommended to play around with things and restart if needed. Trying to route a poorly placed board is a pain

For traces where it matters (high frequency and high speed signals), make sure to use appropriate trace widths and clearances for the desired impedance

Use wider traces for power nets. You can use tables/calculators that tell you how wide you need to be for a certain current, copper thickness, and thermal behavior. However, if you have the space it's a good idea to just use it.

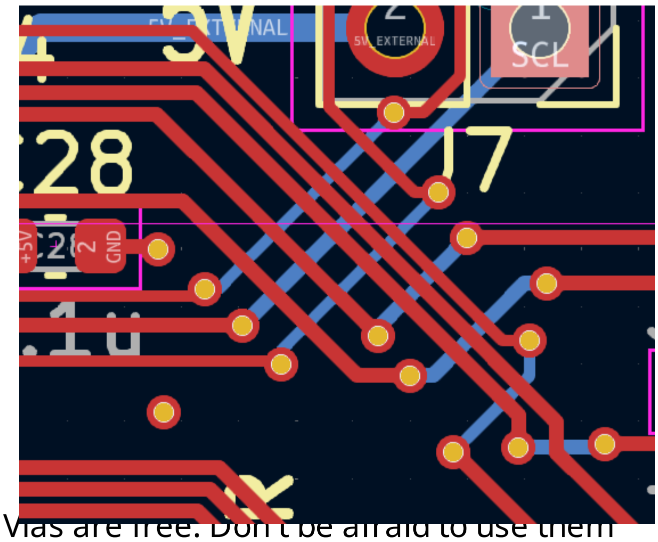
- For power nets it might be useful to route these earlier and to use a tree topology. In other words, route a thick trace into the general area, then break it off into several smaller branches to power all the various things that need power.

Some other random tips:

- Use thicker traces for THT connections. It makes them a bit more resilient to heat

- For two layer boards, use layer as the primary layer (probably the top). Try to conserve most of the bottom layer for ground plane. If you need to use the bottom layer, make short hops under whatever trace you're trying to get around, then return to the top.

- For example, you can setup little cossing zones like this



- Vias are free. Don't be afraid to use them

- Make sure human interfacing things are labelled

- Consider how your board will be mounted