Introduction to KiCad

Here are some notes we made while designing the first printed circuit board.

We chose to use a program called KiCad and most tips are related to this program and its quirks.

Goal was to create a PCB so we could test isolation milling with Proxxon MF70.

General Tips

KiCad uses a lot of popup messages while alarming, these are usually nothing to worry about.

Many functions are accessed with the right mouse button.

- Available options depend on the active tool, tracks can’t be deleted if track tool is not active.

KiCad is actually a collection of small programs.

- Finished project requires the use of three programs and this creates three distinct steps.

If you want to use middle mouse button to pan

- Go to Preferences, Select General or Options, tick “Use middle mouse button to pan”

1. Schematic editor

Passive components can be found under Device category in the Library browser, while ground and voltage are in the Power category.

Shortcut for rotate is R.

After you have placed your components and completed the wiring

- Annotate (top row, next to the ladybug)
- Perform design rules check (DRC, the ladybug) and fix any errors

Create a netlist

Dealing with DRC errors

Every pin has to be connected to something. Unused pins have to be terminated with No connect flag, which can be found on the right hand panel (blue cross)

KiCad requires the use of component called power flag.

- If you don’t place power flags you get the following DRC error “ErrType(3): Pin connected to some others pins but no pin to drive it”
Circuit above is missing power flags so it will give two type 3 errors

Adding two power flags removes the two errors. (power flags can be found in the power category)

"The rules say that a pin on a component that is defined as "power in" must connect to something that is a "power out"*
http://t62146.cad-kicad-user.cadtalk.us/beginner-question-t62146.html

9V is a power input, VI pin on the regulator is a regular input. The connection between 9V and VI therefore needs a power output(PWR_FLAG). Same goes for GND.
5V doesn't need a power flag because 7805 VO is defined as power out and 5V is a power input. This can be checked by editing pin properties in the library editor.

2. Associate schematic with footprints using CvPcb

In this step you decide:

Whether you want to use surface mount or through hole components.
Components footprints, hole sizes and spacings etc.

   Press View selected footprint

   If suitable footprint doesn't exist you have to design your own footprint

3. PCB editor

Start by importing the netlist created with schematic editor.

PCB editor imports all the footprints and places them on a unusable stack.

   Break the stack by activating automatic move

   Then RMB | Glob move and place | Move All Modules

   When done deactivate automatic move

Wiring fine pitch components will require a small grid size

   On the right hand side is the layer list. You can select the working layer and which layers are rendered. Layers include copper, board outline(edges), solder mask, solder paste, silkscreen and adhesive.

   Coordinates (dx,dy) at the bottom can be zeroed with spacebar.

   Drag tries to preserve connections, move breaks connections to component.

   After drawing a copper filled zone only the hatched outline is showed. To visualize the fill use RMB with fill tool active.

   Copper fill has to be removed if you want to add or edit tracks and see the results.

   Solder mask and solder paste clearances can be adjusted under Dimensions menu (top row, next to preferences)

   Remember to place coordinate origins for the drill files. (Right hand panel at the bottom)

Be careful with surface mount devices, it is easy to put them on the wrong side of the PCB.

   It will be hard to connect track to pad if they are not on the same side of the PCB

   To change SMD component from one side to the other

   Click the component with RMB | Footprint | Flip

When changing clearances and track widths with design rules, the changes are not dynamic.

   Removing and reapplying copper fills fixes clearances

   Changes in track width require clicking a track with RMB and selecting Global tracks and vias edition
Pad dimensions and drill diameters are harder to control on a global level. Good library is therefore essential to fast workflow.

Global Pads Edition allows you to open a pad editor which can alter pad size and drill diameter, these changes can then be applied to multiple pads.

Gerbers can be produced with File | Plot

"Use auxiliary axis as origin" is needed for the drill files to have the right coordinate system