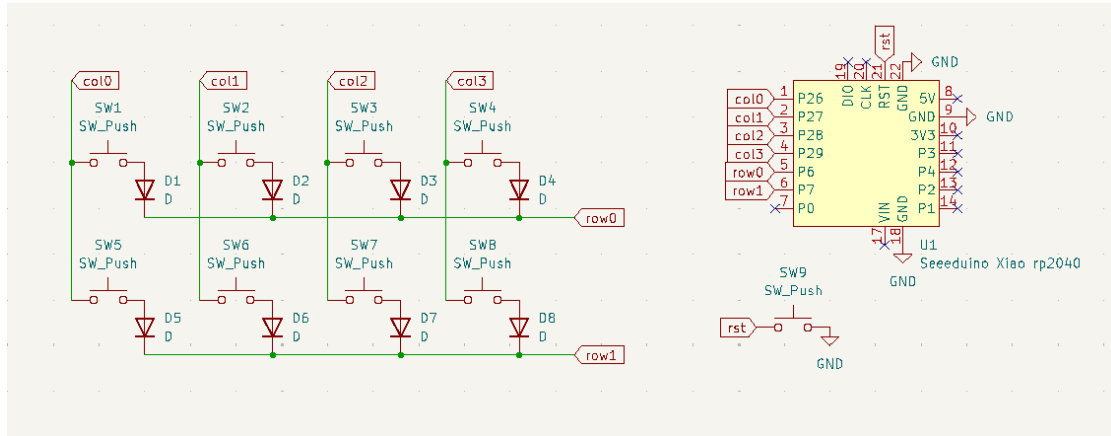


KiCad Workshop

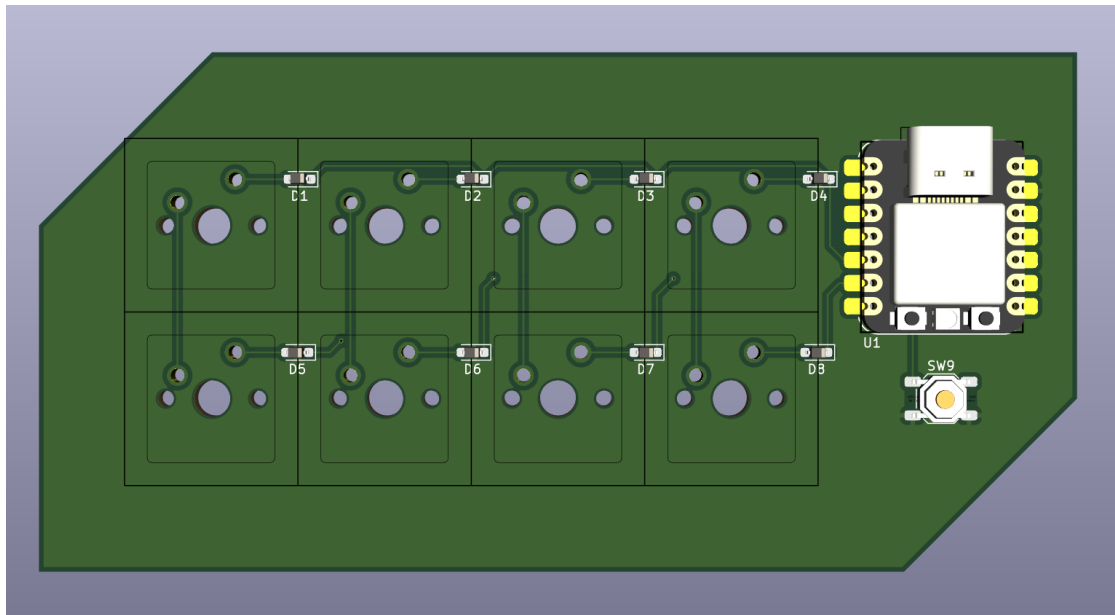
- Create Project
- Install marbastlib <https://github.com/ebastler/marbastlib>

Schematic



- mouse select from left, everything that is completely in selection area is included in selection
- mouse select from right, everything that touches selection are included in selection
- a place part
- w add wire
- r rotate
- m move component
- del delete
- g grab, same as m but move wires with it
- l/ctrl-l add label/global-label
- q mark pins not connected
- p add power net
- v edit value
- e edit properties
- assign footprints
- F8 update PCB from schematic

PCB



- set custom grid, alternate between grids `alt-1`, `alt-2`
- set shove setting for traces
- `m` move
- `r` rotate, `R` rotate clockwise
- `x` lay down trace
- `v` add via (in trace command)
- `f` finish trace (in trace command)
- `u` select part of trace (in trace command), multiple presses select more
- `v` (not in trace) flip board
- `f` flip component to other side
- `d` displace (move with traces)
- `b` update fills
- `h` hide/dim other stuff
- DRC, design rule checker
- edge cut
- `alt-3` view 3d render
- add fill layer

PCB-art

- <https://www.youtube.com/watch?v=Stp5BqUI48U> Kliment's PCB art for terrified beginners workshop