Design a Circuit Board with KiCad

Schematic

- Create a new Project and open the Schematic Editor
- Add components with a and power-ports with p (keybindings)
- Mouse over parts and move, rotate and flip (x, y) them
- Wire up parts w, and undo fixed segments with Backspace
- Set values v and run the Footprint assignment tool
- Annotate Schematic with Reference Designators
- Run ERC, fix all errors and understand all warnings

Board

- Open the PCB Editor and Update the PCB
- Select a Grid size n, Shift-n; set the Interactive Router to shove
- Move parts with m, flip to bottom, rotate
- Route tracks with x, place vias and switch active layer with v
- Delete track segments and Shift-Del entire tracks
- Create a board outline on the *Edge.Cuts* layer
- Create a copper *filled zone*, fill it with b
- Cleanup Silkscreen layers
- Run DRC, fix errors and understand warnings
- Generate Gerber files, inspect them with a Gerber Viewer

Hints

- Exit a tool with Esc (possibly multiple times)
- e to edit an object works almost everywhere
- This workflow is just one, there are many ways to archive the same result
- Some distributions have extra packages for symbols, footprints and 3D models. Install them.

Schematic "Astable multivibrator"
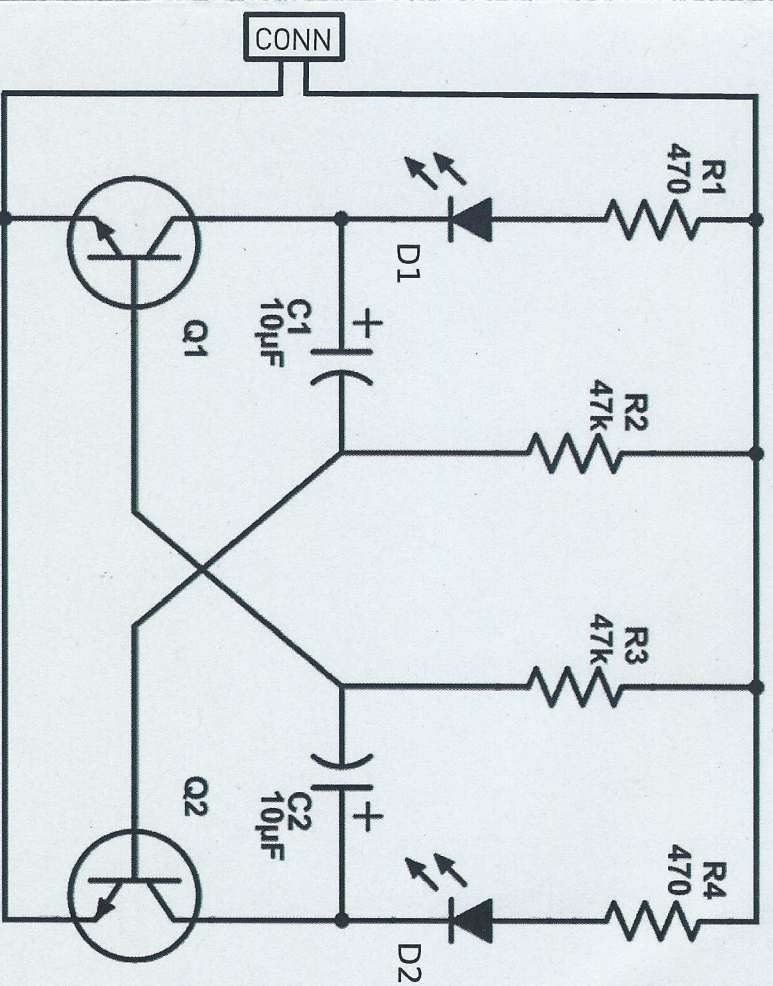
Matching KiCad Symbols to [Footprints]

- R (Resistor R1..R4) [R_0805...]
- LED (D1, D2) [LED_0805...]
- C_Polarized (Capacitor C1, C2) [Capacitor_Tant...:CP_EIA-3528-15...]
- BC817 (Q1, Q2) (fully specified – means it comes with a footprint)
- Conn_01x02 (power connector) [Connec...2.54:PinHeader_1x02_P2.54]