

# Design a Circuit Board with KiCad

## Schematic

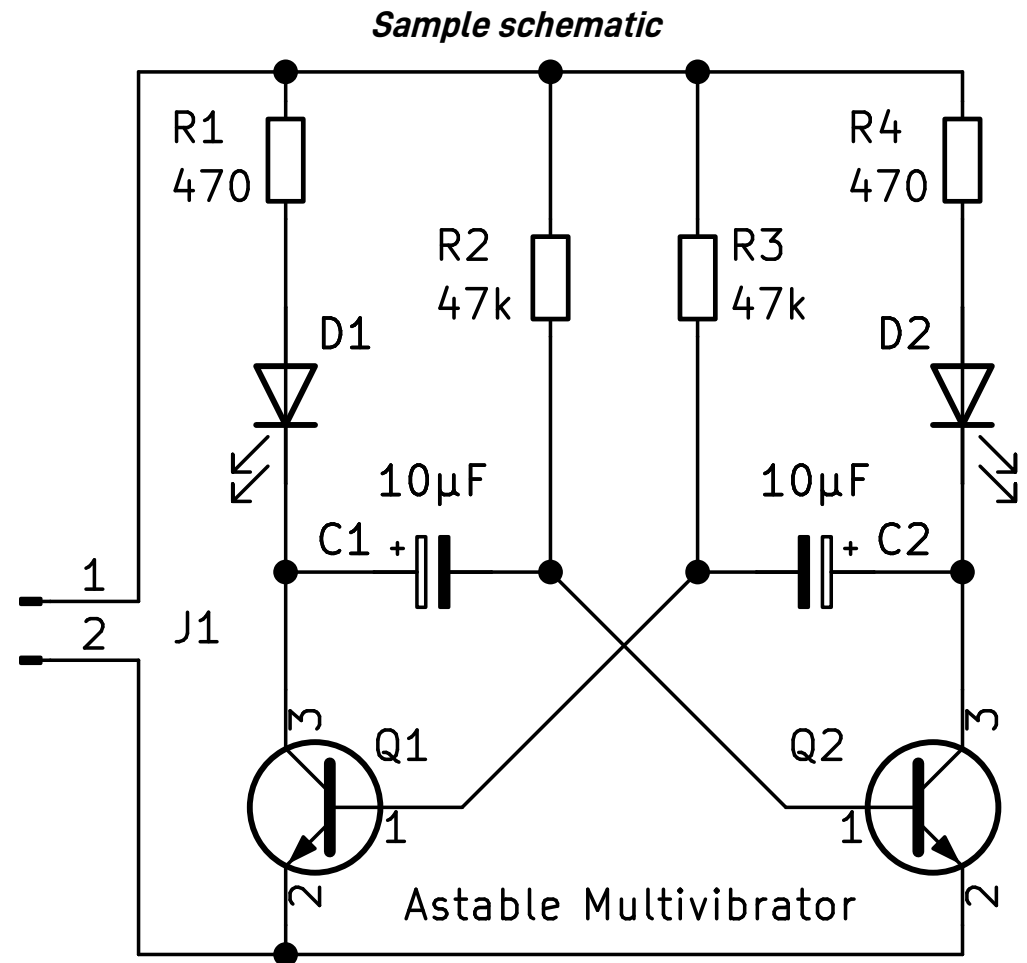
- Create a new Project and open the Schematic Editor
- Add components with **a** and power-ports with **p** (keybindings)
- Mouse over parts and **move**, **rotate** and **flip (x, y)** them
- Wire up parts **w**, and undo fixed segments with **Backspace**
- Set values **v** and run the Footprint assignment tool
- 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**
- Attempt finishing a track in progress or **Shift+f** for several pads
- **Delete** track segments and **Shift-Del** entire tracks
- Create a board outline on the *Edge.Cuts* layer
- Run DRC, fix errors and understand warnings
- Clean up Silkscreen layers
- 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.



## Matching KiCad Symbols to [Footprints]

- **R** (Resistor R1..R4) [**R\_0805...**]
- **LED** (D1, D2) [**LED\_0805...**]
- **C\_Polarized** (Capacitor C1, C2) [**CP\_Radial\_D5.0mm\_P2.00mm**]
- **BC817** (Q1, Q2) (fully specified – means it comes with a footprint)
- **Conn\_01x02** (J1 (pins 1 and 2)) [**Connec...2.54:PinHeader\_1x02\_P2.54**]