

Design a Circuit Board with KiCad

Workflow

- Create new Project, open schematic-editor
- Add components **a** and power-ports **p**
- Move parts **m**, rotate **r**, flip **x y**
- Wire up parts **w**
- Assign footprints **f**, set value **v**
- Annotate Schematic, switch to board
- Update board from schematic
- Select a grid **n**, **SHIFT-n**
- Create outline on *Edge.Cuts* layer
- Move parts **m**, flip to bottom **f**, rotate **r**
- Route tracks **x**, rip-up **DEL**, via **v**

Workflow Part2

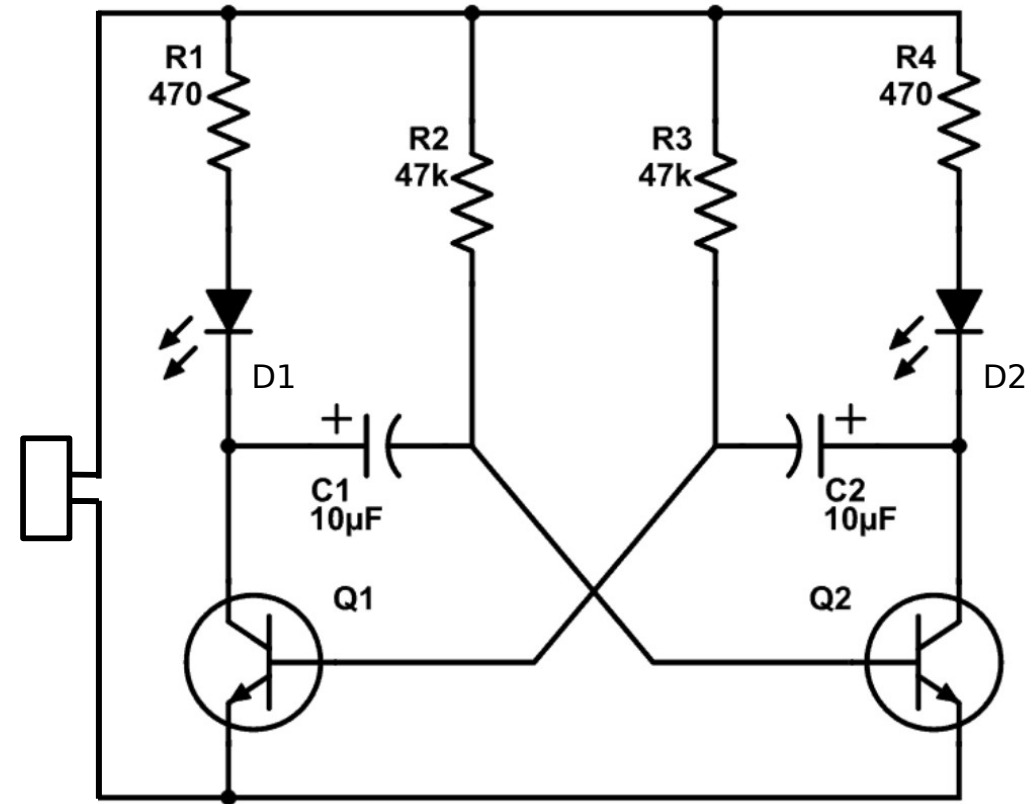
- Create copper fill
- Cleanup Silkscreen and Fab layers
- Run DRC, fix errors and warnings
- Export gerber-files, inspect them

Hints

- Exit a tool with **ESC**
- **e** to edit an object works almost everywhere
- This workflow is just one of many, there are so many ways to archive the same result
- Some Linux distributions have extra packages for symbols, footprints and 3D-Models. Install them.

Steam: <https://at.omcat.info>

Schematic “Astable multivibrator”



Matching KiCad Symbols [Footprints]

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

Design a Circuit Board with KiCad

Workflow

- Create new Project, open schematic-editor
- Add components **a** and power-ports **p**
- Move parts **m**, rotate **r**, flip **x y**
- Wire up parts **w**
- Assign footprints **f**, set value **v**
- Annotate Schematic, switch to board
- Update board from schematic
- Select a grid **n**, **SHIFT-n**
- Create outline on *Edge.Cuts* layer
- Move parts **m**, flip to bottom **f**, rotate **r**
- Route tracks **x**, rip-up **DEL**, via **v**

Workflow Part2

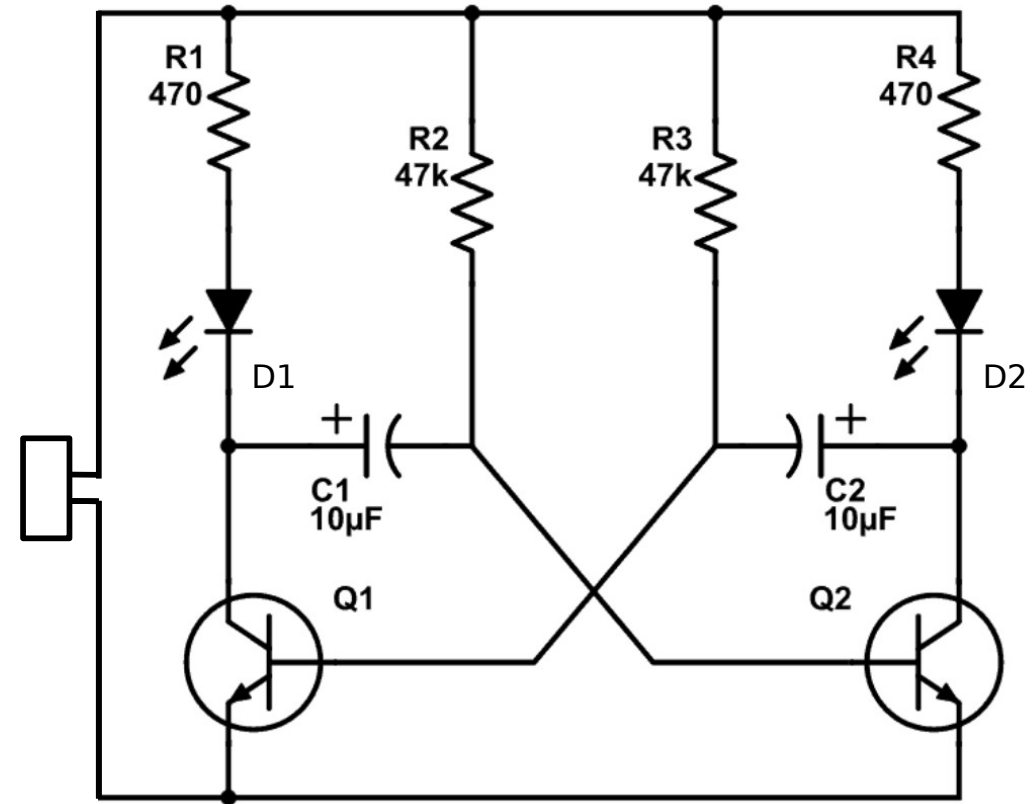
- Create copper fill
- Cleanup Silkscreen and Fab layers
- Run DRC, fix errors and warnings
- Export gerber-files, inspect them

Hints

- Exit a tool with **ESC**
- **e** to edit an object works almost everywhere
- This workflow is just one of many, there are so many ways to archive the same result
- Some Linux distributions have extra packages for symbols, footprints and 3D-Models. Install them.

Steam: <https://at.omcat.info>

Schematic “Astable multivibrator”



Matching KiCad Symbols [Footprints]

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