

KiCad v5.0.1 cheatsheet

<http://kicad-pcb.org/help/documentation/>

1) Create a project

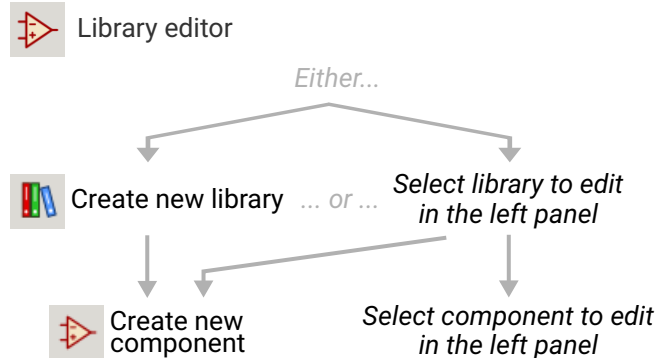
File → New Project → New Project

2) Eeschema : draw the schematic

Add components : [A]
 Move item¹ : [M]
 Grab item¹ : [G]
 Duplicate item : [C]
 Duplicate selection : [Shift] + []
 Delete item : [Del]
 Delete selection : [Ctrl] + [Shift] + []
 Rotate item : [R]
 Mirror item : [X] / [Y]
 Add wires : [W]
 Edit properties : [E]
 Edit value : [V]
 Add power symbols : [P]
 Add no-connect : [Q]
 Add text : [T]
 Add labels : [L]
 List of shortcuts : [Ctrl] + [F1]

¹grab keeps connections, move doesn't

3) Create new components as necessary



Draw component

Add pins : [P]

Repeat pin : [Ins]

Most shortcuts are identical to Eeschema

Save all new/modified libraries : [Ctrl] + [S]

4) Create and assign footprints

Footprint Editor

If editing an existing library : Select active library

New footprint / Load footprint from library

Draw component

Add pins

Save footprint in active library / Create new library and save current footprint

Run CvPcb to associate components and footprints

How to load the new library in CvPcb :

Preferences → Footprint libraries

Append with wizard

Select your .pretty folder

5) Pcbnew : design the layout

Design Rules → Design Rules + Layers Setup

Tools → Update PCB from schematic...

Repeat every time the schematic is modified

Select top layer : [PgUp]

Select bottom layer : [PgDn]

Move item¹ : [M]

Grab item¹ : [G]

Duplicate item : [C]

Rotate item : [R]

Add tracks : [X]

Add via : [V]

Switch posture : [Q]

Switch track width : [W]

Drag track : [D]

Fill zones : [B]

3D viewer : [Alt] (+ [Shift]) + [3]

¹grab keeps connections, move doesn't (Only for AZERTY keyboards)

6) Export Gerbers

File → Plot

Generate Drill File + Plot

Check result using GerbView