KiCad 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 item[1] : + M
- Grab item[1] : + G
- Copy item : + C
- Copy 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 : ?

[1]*grab* keeps connections, *move* doesn't

3) Create new components as necessary

→ Library editor

If editing an existing library : Select working library

Create new component / Load component to edit from current library

Draw component

Add pins ..... P

Update current component into current library / Save current component to new library

How to load the new library in Eeschema :

Preferences → Component libraries

Component library files → Add

Select your `.lib` *file*

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*

→ Generate netlist

5) Pcbnew : design the layout

Design Rules → Design Rules + Layers Setup

→ Read netlist

- Move item[1] : + M
- Grab item[1] : + G
- Copy 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

[1]*grab* keeps connections, *move* doesn't

(Only for AZERTY keyboards)

6) Export Gerbers

File → Plot

Generate Drill File + Plot → Check result using GerbView