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

Select top layer :	⬆ PgUp
Select bottom layer :	⬇ PgDn
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