KiCad Cheatsheet

https://docs.kicad.org

1) Create a project

File → New Project → New Project

2) Schematic Editor

Add components :	A
Move item[1] :	+ M
Grab item[1] :	+ G
Expand selection:	⇧ Shift +
Deselect items:	Ctrl + ⇧ Shift +
Delete item :	+ Del
Edit Symbol :	+ Ctrl + E
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 + ⇧ Shift + F1

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

3) Create new symbols as necessary

→ Library editor

Create new symbol / Import symbol to edit from file

Draw symbol elements

Add pins .... P

Edit pins

Set Symbol Properties

Save symbol in library / Export current symbol to schematic

How to load the new library in Library Editor :

File → Add Library

-or-

Preferences → Manage Symbol libraries

Add Existing Library

4) Create and assign footprints

→ Footprint Editor

New footprint / New Footprint Wizard

Draw elements

Add pads

Set Footprint Options

Save footprint in active library

How to assign footprint in Schematic Editor :

Edit Symbol Fields

-or-

Assign Footprints

5) PCB Editor

→ Update PCB From Schematic

→ Board Setup

Switch Viewport	⇧ Shift + Tab
Switch Active Layerset	Ctrl + Tab
Move item :	+ M
Flip item side :	+ F
Rotate item :	+ R
Add footprint :	A
Add tracks :	X
Add via[1] :	Ctrl + ⇧ Shift + V
Switch posture :	/
Switch track width :	W
Drag track / footprint :	D
Fill zones :	B
3D viewer :	Alt + 3
Measure :	Ctrl + ⇧ Shift + M

[1] while routing, only 'V' is needed

6) Export Gerbers / IPC2581

File → Fabrication Outputs

.gbr / .xml

→ Check result with Gerber Viewer