

KiCad 5.0.2 cheat sheet

EE Schema

?	Help
F1	Zoom in
F2	Zoom out
F3	Zoom redraw
C	Duplicate component or label
A	Add component
R	Rotate item
X	Mirror X Component
Ctrl + E	Edit with Symbol Editor
W	Begin wire
H	Add hierarchical label
F8	Update PCB from Schematic
O	Autoplace fields
Backspace	Delete node
Ctrl + B	Highlight connection

Pcbnew

F1	Zoom in
S	Set Grid Origin
/	Switch Track Posture
`	Toggle Highlight of Selected Net (Modern Toolset only)
F11	Switch to Modern Toolset with hardware-accelerated graphics (recommended)

Last modification: 7/17/2020 10:21:48 AM

More information: defkey.com/kicad-shortcuts?filter=basic

[Customize this PDF...](#)