

KiCAD Hotkeys

EESCHEMA

Help	?
Zoom In	F1
Zoom Out	F2
Zoom Redraw	F3
Zoom Center	F4
Fit on Screen	Home
Delete Item	Del
Repeat Lest Item	Ins
Move Block -> Drag Block	Tab
Move Item	M
Copy Component or Label	C
Drag Item	G
Add Component	A
Add Power	P
Rotate Item	R
Mirror X Component	X
Mirror Y Component	Y
Orient Normal Component	N
Edit Item	E
Edit Value	V
Edit Reference	U
Edit Footprint	F
Begin Wire	W
Begin Bus	B
End Line Wire Bus	K
Add Label	L
Add Hierarchical Label	H
Add Junction	J
Add No Connect Flag	Q
Add Sheet	S
Add Wire Entry	Z
Add Bus Entry	/

PCBNEW

Switch Units	CTRL+U
Reset Local Coordinates	Space
Undo	Ctrl+Z
Redo	CTRL-Y
Track Display Mode	K
Delete	Del
Delete Track Segment	BkSp
Add New Track	X
Add Via	V
Add Microvia	CTRL+V
Switch Track Posture	/
Drag Track, Keep Slope	D
Place Item	P
Copy Item	C
End Track	End
Move Item	M
Flip Item	F
Rotate Item	R
Drag Item	G
Get and Move Footprint	T
Lock/Unlock Footprint	L
Save Board	Ctrl+S
Load Board	Ctrl+L
Find Item	Ctrl+F
Edit Item	End
Switch to Cooper Layer	PgDn
Switch to Inner Layer 1	F5
Switch to Inner Layer 2	F6
Switch to Component Layer	PgUp
Switch to Next Layer	+
Switch to Previous Layer	-
Add Module	O
Switch Track Width to Next	W
Switch Track Width to Previous	Ctrl+W
Switch Highcontrast Mode	H
Invert (mirror) Block	Shift-LeftMouseButton