KiCAD Hotkeys

EESCHEMA		PCBNEW	
Help	?	Switch Units	CTRL+U
Zoom In	F1	Reset Local Coordinates	Space
Zoom Out	F2	Undo	Ctrl+Z
Zoom Redraw	F3	Redo	CTRL-Y
Zoom Center	F4	Track Display Mode	K
Fit on Screen	Home	Delete	Del
Delete Item	Del	Delete Track Segment	BkSp
Repeat Lest Item	Ins	Add New Track	X
Move Block -> Drag Block	Tab	Add Via	V
Move Item	M	Add Microvia	CTRL+V
Copy Component or Label	C	Switch Track Posture	/
Drag Item	G	Drag Track, Keep Slope	D
Add Component	A	Place Item	P
Add Power	P	Copy Item	C
Rotate Item	R	End Track	End
Mirror X Component	X	Move Item	M
Mirror Y Component	Y	Flip Item	F
Orient Normal Component	N	Rotate Item	R
Edit Item	E	Drag Item	G
Edit Value	V	Get and Move Footprint	T
Edit Reference	U	Lock/Unlock Footprint	L
Edit Footprint	F	Save Board	Ctrl+S
Begin Wire	W	Load Board	Ctrol+L
Begin Bus	B	Find Item	Ctrl+F
End Line Wire Bus	K	Edit Item	End
Add Label	L	Switch to Cooper Layer	PgDn
Add Hierarchical Label	H	Switch to Inner Layer 1	F5
Add Junction	J	Switch to Inner Layer 2	F6
Add No Connect Flag	Q	Switch to Component Layer	PgUp
Add Sheet	S	Switch to Next Layer	+
Add Wire Entry	Z	Switch to Previous Layer	-
Add Bus Entry	/	Add Module	O
		Switch Track Width to Next	W
		Switch Track Width to Previous	Ctrl+W
		Switch Highcontrast Mode	H
		Invert (mirror) Block	Shift-LeftMouseB