



KiCad 5.0.2 cheat sheet

EE Schema

?	Help
F1	Zoom in
F2	Zoom out
F3	Zoom redraw
F4	Zoom center
Home	Fit on screen
Del	Delete item
Insert	Repeat last item
Tab	Move block > Drag block
M	Move schematic item
C	Duplicate component or label
G	Drag item
A	Add component
P	Add power
R	Rotate item
X	Mirror X Component
Y	Mirror Y Component
N	Orient normal component
E	Edit item
V	Edit value
U	Edit reference
F	Edit footprint
Ctrl + E	Edit with Symbol Editor
W	Begin wire
B	Begin bus
K	End line wire bus
L	Add label
H	Add hierarchical label
Ctrl + H	Add global label
J	Add junction
Q	Add no connect flag
S	Add sheet
Z	Add wire entry
/	Add bus entry
Enter	Left click
End	Double click
Ctrl + F	Find item
F5	Find next item

Shift + F5 Find next DRC marker

Ctrl + Alt + F Find and replace

I Add graphic PolyLine

T Add Graphic Text

F8 Update PCB from Schematic

O Autoplace fields

Alt + Backspace Leave sheet

Backspace Delete node

Ctrl + B Highlight connection

Pcbnew

Ctrl + O Open

Ctrl + S Save

Ctrl + Shift + S Save As

Ctrl + P Print

Ctrl + Z Undo

Ctrl + Y Redo

Ctrl + X Cut

Ctrl + C Copy

Ctrl + V Paste

Ctrl + F1 Help (this window)

F1 Zoom in

F2 Zoom out

F3 Zoom Redraw

F4 Zoom center

Home Zoom Auto

Ctrl + F5 Zoom to Selection

Alt + 3 3D Viewer

Ctrl + U Switch Units

Space Reset Local Coordinates

S Set Grid Origin

Z Reset Grid Origin

Enter Mouse Left Click

End Mouse Left Double Click

} Increment Layer Transparency (Modern Toolset only)

{ Decrement Layer Transparency (Modern Toolset only)

Ctrl + Shift + X Toggle Cursor Display (Modern Toolset only)

Ctrl + Shift + M	Measure Distance (Modern Toolset only)
P	Place Item
O	Add Footprint
Ctrl + Shift + V	Add Vias
Ctrl + Shift + Z	Add Filled Zone
Ctrl + Shift + K	Add Keepout Area
C	Add a Zone Cutout
Ctrl + Shift + .	Add a Similar Zone
Ctrl + Shift + L	Draw Line
Ctrl + Shift + C	Draw Circle
Ctrl + Shift + A	Draw Arc
Ctrl + Shift + P	Draw Graphic Polygon
Ctrl + Shift + T	Add Text
Ctrl + Shift + H	Add Dimension
Ctrl + Shift + F	Place DXF
X	Add New Track
/	Switch Track Posture
D	Drag Track Keep Slope
V	Add Through Via
Alt + Shift + V	Add Blind/Buried Via
Ctrl + V	Add MicroVia
6	Route Differential Pair (Modern Toolset only)
7	Tune Single Track (Modern Toolset only)
8	Tune Differential Pair Length (Modern Toolset only)
9	Tune Differential Pair Skew (Modern Toolset only)
Ctrl + ,	Routing Options
Ctrl + L	Length Tuning Settings (Modern Toolset only)
1	Increase meander spacing by one step.
2	Decrease meander spacing by one step.
3	Increase meander amplitude by one step.
4	Decrease meander amplitude by one step.
Ctrl + Shift + R	Differential Pair Dimensions

B	Fill or Refill All Zones
Ctrl + B	Remove Filled Areas in All Zones
U	Select Trivial Connection
I	Select Copper Connection
`	Toggle Highlight of Selected Net (Modern Toolset only)
K	Track Display Mode
Q	Custom Track/Via Size
W	Switch Track Width To Next
Ctrl + W	Switch Track Width To Previous
'	Increase Via Size
\	Decrease Via Size
Del	Delete Track or Footprint
Back	Delete Track Segment
G	Drag Item
C	Copy Item
M	Move Item
T	Get and Move Footprint
Ctrl + M	Move Item Exactly
Ctrl + R	Position Item Relative
F	Flip Item
R	Rotate item
Shift + R	Rotate Item Clockwise (Modern Toolset only)
<	Select Layer and Add Through Via
Alt + <	Select Layer and Add Blind/Buried Via
Ctrl + D	Duplicate Item
Ctrl + Shift + D	Duplicate Item and Increment
Ctrl + T	Create Array
L	Lock/Unlock Footprint
Ctrl + F	Find item
E	Edit Item
Ctrl + E	Edit with Footprint Editor
Page Up	Switch to Component (F.Cu) layer
Page Down	Switch to Copper (B.Cu) layer
F5	Switch to Inner layer 1
F6	Switch to Inner layer 2

F7	Switch to Inner layer 3
F8	Switch to Inner layer 4
Shift + F5	Switch to Inner layer 5
Shift + F6	Switch to Inner layer 6
+	Switch to Next Layer
-	Switch to Previous Layer
Alt + 1	Switch Grid To Fast Grid1
Alt + 2	Switch Grid To Fast Grid2
N	Switch Grid To Next
Shift + N	Switch Grid To Previous
H	Toggle High Contrast Mode
F9	Switch to Legacy Toolset (not all features will be available)
F12	Switch to Modern Toolset with software graphics (fall-back)
F11	Switch to Modern Toolset with hardware-accelerated graphics (recommended)

KiCad Manager

Ctrl + N	New project
Ctrl + T	New project from template
Ctrl + O	Open project
Ctrl + R	Refresh project tree
Ctrl + F1	Help (this window)
Ctrl + E	Run Eeschema
Ctrl + L	Run LibEdit
Ctrl + P	Run Pcbnew
Ctrl + F	Run FpEditor
Ctrl + G	Run Gerbview
Ctrl + B	Run Bitmap2Component
Ctrl + A	Run PcbCalculator
Ctrl + Y	Run PLEditor

GerbView

Ctrl + F1	Help (this window)
F1	Zoom in
F2	Zoom out
F3	Zoom redraw
F4	Zoom center

Home	Zoom Auto
Ctrl + F5	Zoom to Selection
U	Switch units
Space	Reset Local Coordinates
L	Gbr lines display mode
F	Gbr flashed display mode
P	Gbr polygons display mode
D	D Codes display mode
N	Gbr negative obj display mode
+	Switch to next layer
-	Switch to previous layer
F9	Switch to legacy toolset
F11	Switch to modern toolset with hardware-accelerated graphics (recommended)
F12	Switch to modern toolset with software graphics (fall-back)
Ctrl + Shift + M	Measure distance (modern toolset only)

Page Layout Editor

Ctrl + N	New
Ctrl + O	Open
Ctrl + S	Save
Ctrl + Shift + S	Save As
Ctrl + P	Print
Ctrl + Z	Undo
Ctrl + Y	Redo
F1	Zoom in
F2	Zoom out
F3	Zoom redraw
F4	Zoom center
Home	Zoom Auto
Ctrl + F5	Zoom to Selection
Space	Reset Local Coordinates
Ctrl + F1	Help (this window)
Enter	Left click
End	Double click
M	Move Item
S	Move start point

E	Move end point
P	Place Item
Del	Delete item

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

More information: defkey.com/ha/kicad-shortcuts

[Customize this PDF...](#)