

6 Command-Line Options

The synopsis of the `pcb` command is:

```
pcb [OPTION ...] [LAYOUT-FILE.pcb] to start the application in GUI mode,
or
pcb [-h | -V | --copyright] for a list of options, version, and copyright,
or
pcb -p [OPTION ...] [LAYOUT-FILE.pcb] to print a layout,
or
pcb -x HID [OPTION ...] [LAYOUT-FILE.pcb] to export.
```

Possible values for the parameter ‘HID’ are:

```
‘bom’      Export a bill of materials.
‘gerber’   RS-274X (Gerber) export.
‘nelma’    Numerical analysis package export.
‘png’      GIF/JPEG/PNG export.
‘ps’       Postscript export.
‘eps’      Encapsulated Postscript
```

There are several resources which may be set or reset in addition to the standard toolkit command-line options. For a complete list refer to [Section 7.1 \[Resources\]](#), page 46.

6.1 bom-Options

```
--bomfile <string>
    Name of the BOM output file.
--xyfile <string>
    Name of the XY output file.
--xy-in-mm
    XY dimensions in mm instead of mils.
```

6.2 Color-Options

```
--black-color <string>
    Color value for black. Default: ‘#000000’
--white-color <string>
    Color value for white. Default: ‘#ffffff’
--background-color <string>
    Color of the background of the canvas. Default: ‘#e5e5e5’
--crosshair-color <string>
    Color of the crosshair. Default: ‘#ff0000’
```

```
--cross-color <string>
    Color of the cross. Default: '#cdcd00'

--via-color <string>
    Color of vias. Default: '#7f7f7f'

--via-selected-color <string>
    Color of selected vias. Default: '#00ffff'

--pin-color <string>
    Color of pins. Default: '#4d4d4d'

--pin-selected-color <string>
    Color of selected pins . Default: '#00ffff'

--pin-name-color <string>
    Color of pin names and pin numbers. Default: '#ff0000'

--element-color <string>
    Color of components. Default: '#000000'

--rat-color <string>
    Color of ratlines. Default: '#b8860b'

--invisible-objects-color <string>
    Color of invisible objects. Default: '#cccccc'

--invisible-mark-color <string>
    Color of invisible marks. Default: '#cccccc'

--element-selected-color <string>
    Color of selected components. Default: '#00ffff'

--rat-selected-color <string>
    Color of selected rats. Default: '#00ffff'

--connected-color <string>
    Color to indicate connections. Default: '#00ff00'

--off-limit-color <string>
    Color of off-canvas area. Default: '#cccccc'

--grid-color <string>
    Color of the grid. Default: '#ff0000'

--layer-color-<n> <string>
    Color of layer <n>, where <n> is an integer from 1 to 16.

--layer-selected-color-<n> <string>
    Color of layer <n>, when selected. <n> is an integer from 1 to 16.

--warn-color <string>
    Color of offending objects during DRC. Default value is "#ff8000"

--mask-color <string>
    Color of the mask layer. Default value is "#ff0000"
```

6.3 Command-Options

pcb uses external commands for input output operations. These commands can be configured at start-up to meet local requirements. The command string may include special sequences %f, %p or %a. These are replaced when the command is called. The sequence %f is replaced by the file name, %p gets the path and %a indicates a package name.

```
--font-command <string>
    Command to load a font.

--file-command <string>
    Command to read a file.

--element-command <string>
    Command to read a footprint. Defaults to "M4PATH='%p' ; export M4PATH; echo
    'include(%f)' | m4"

--print-file <string>
    Command to print to a file.

--lib-command-dir <string>
    Path to the command that queries the library.

--lib-command <string>
    Command to query the library. Defaults to "QueryLibrary.sh '%p' '%f' %a"

--lib-contents-command <string>
    Command to query the contents of the library. Defaults to
    "ListLibraryContents.sh %p %f"

--save-command <string>
    Command to save to a file.

--rat-command <string>
    Command for reading a netlist. Sequence %f is replaced by the netlist filename.
```

6.4 DRC-Options

The unit of all design rule options (DRC) is 1/100 mil.

```
--bloat <num>
    Minimum spacing. Default value is 1000

--shrink <num>
    Minimum touching overlap. Default value is 1000

--min-width <num>
    Minimum width of copper. Default value is 1000

--min-silk <num>
    Minimum width of lines in silk. Default value is 1000

--min-drill <num>
    Minimum diameter of holes. Default value is 100

--min-ring <num>
    Minimum width of annular ring. Default value is 1000
```

6.5 eps-Options

- `--eps-file <string>`
Name of the encapsulated postscript output file. Can contain a path.
- `--eps-scale <num>`
Scale EPS output by the parameter 'num'.
- `--as-shown`
Export layers as shown on screen.
- `--monochrome`
Convert output to monochrome.
- `--only-visible`
Limit the bounds of the EPS file to the visible items.

6.6 gerber-Options

- `--gerberfile`
Gerber output file prefix. Can include a path.
- `--all-layers`
Output contains all layers, even empty ones.
- `--verbose`
Print file names and aperture counts on stdout.

6.7 GUI-Options

- `--pinout-offset-x <num>`
Horizontal offset of the pin number display in mil. Defaults to 100
- `--pinout-offset-y <num>`
Vertical offset of the pin number display in mil. Defaults to 100
- `--pinout-text-offset-x <num>`
Horizontal offset of the pin name display in mil. Defaults to 800
- `--pinout-text-offset-y <num>`
Vertical offset of the pin name display in mil. Defaults to -100
- `--draw-grid <string>`
If set, draw the grid at start-up.
- `--clear-line`
If set, new lines clear polygons.
- `--full-poly`
If set, new polygons are full ones.
- `--unique-names`
If set, you will not be permitted to change the name of an component to match that of another component.

- `--snap-pin`
If set, pin centers and pad end points are treated as additional grid points that the cursor can snap to.
- `--all-direction-lines`
Allow all directions, when drawing new lines.
- `--show-number`

6.8 Layername-Options

- `--layer-name-1 <string>`
Name of the 1st Layer. Default is "component".
- `--layer-name-2 <string>`
Name of the 2nd Layer. Default is "solder".
- `--layer-name-3 <string>`
Name of the 3rd Layer. Default is "GND".
- `--layer-name-4 <string>`
Name of the 4th Layer. Default is "power".
- `--layer-name-5 <string>`
Name of the 5th Layer. Default is "signal1".
- `--layer-name-6 <string>`
Name of the 6th Layer. Default is "signal2".
- `--layer-name-7 <string>`
Name of the 7th Layer. Default is "signal3".
- `--layer-name-8 <string>`
Name of the 8th Layer. Default is "signal4".

6.9 lpr-Options

- `--lprcommand <string>`
Command to use for printing. Defaults to `lpr`. This can be used to produce PDF output with a virtual PDF printer (E.g. `--lprcommand "lp -d CUPS-PDF-Printer"`).

In addition, all ps-Options are valid.

6.10 nelma-Options

- `--basename <string>`
File name prefix.
- `--dpi <num>`
Horizontal scale factor (grid points/inch).
- `--copper-height <num>`
Copper layer height (um).

`--substrate-height <num>`
Substrate layer height (um).

`--substrate-epsilon <num>`
Substrate relative epsilon.

6.11 Options

`--help` Show help on command line options.

`--version`
Show version.

`--verbose`
Be verbose on stdout.

`--copyright`
Show copyright.

`--show-defaults`
Show option defaults.

`--show-actions`
Show available actions and exit.

`--dump-actions`
Dump actions (for documentation).

`--grid-units-mm`
Grid units are metric on start-up.

`--backup-interval`
Time between automatic backups in seconds. Set to 60 to disable. The default value is 60.

`--groups <string>`
Layer group string. Defaults to "1,c:2,s:3:4:5:6:7:8".

`--route-styles <string>`
A string that defines the route styles. Defaults to "Signal,1000,3600,2000,1000:Power,2500,6000,3000,1000:Fat,4000,6000,3500,1000:Skinny,600,2402,1181,600"

`--element-path <string>`
A colon separated list of directories or commands (starts with '|'). The path is passed to the program specified in '`--element-command`'.

`--menu-file <string>`
Name of the file that defines the menu with the lesstif GUI. Defaults to '`pcb-menu.res`'.

`--action-script <string>`
If set, this file is executed at startup.

`--action-string <string>`
If set, this string of actions is executed at startup.

- fab-author**
Name of author to be put in the Gerber files.
- layer-stack <string>**
Initial layer stackup, for setting up an export. A comma separated list of layer names, layer numbers and layer groups.
- save-last-command**
If set, the last user command is saved.
- save-in-tmp**
If set, all data which would otherwise be lost are saved in a temporary file `‘/tmp/PCB.%i.save’`. Sequence `‘i’` is replaced by the process ID.
- reset-after-element**
If set, all found connections are reset before a new component is scanned.
- ring-bell-finished**
Execute the bell command when all rats are routed.

6.12 Options.

- listen** Listen for actions on stdin.
- bg-image <string>**
File name of an image to put into the background of the GUI canvas. The image must be a color PPM image, in binary (not ASCII) format. It can be any size, and will be automatically scaled to fit the canvas.
- pcb-menu <string>**
Location of the `‘gpcb-menu.res’` file which defines the menu for the GTK GUI.

6.13 Path-Options

- lib-newlib <string>**
Top level directory for the newlib style library.
- lib-name <string>**
The default filename for the library.
- default-font <string>**
The name of the default font.
- file-path <string>**
A colon separated list of directories or commands (starts with `‘|’`). The path is passed to the program specified in `fileCommand` together with the selected filename.
- font-pat <string>**
A colon separated list of directories to search the default font. Defaults to the default library path.
- lib-path <string>**
A colon separated list of directories that will be passed to the commands specified by `elementCommand` and `elementContentsCommand`.

6.14 PNG-Options

- `--outfile <string>`
Name of the file to be exported to. Can contain a path.
- `--dpi` Scale factor in pixels/inch. Set to 0 to scale to size specified in the layout.
- `--x-max` Width of the png image in pixels. No constraint, when set to 0.
- `--y-max` Height of the png output in pixels. No constraint, when set to 0.
- `--xy-max` Maximum width and height of the PNG output in pixels. No constraint, when set to 0.
- `--as-shown`
Export layers as shown on screen.
- `--monochrome`
Convert output to monochrome.
- `--only-vivable`
Limit the bounds of the exported PNG image to the visible items.
- `--use-alpha`
Make the background and any holes transparent.
- `--format <string>`
File format to be exported. Parameter `<string>` can be 'PNG', 'GIF', or 'JPEG'.
- `--png-bloat <num><dim>`
Amount of extra thickness to add to traces, pads, or pin edges. The parameter '`<num><dim>`' is a number, appended by a dimension 'mm', 'mil', or 'pix'. If no dimension is given, the default dimension is 1/100 mil.
- `--photo-mode`
Export a photo realistic image of the layout.
- `--photo-flip-x`
In photo-realistic mode, export the reverse side of the layout. Left-right flip.
- `--photo-flip-y`
In photo-realistic mode, export the reverse side of the layout. Up-down flip.

6.15 ps-Options

- `--psfile <string>`
Name of the postscript output file. Can contain a path.
- `--drill-helper`
Print a centering target in large drill holes.
- `--align-marks`
Print alignment marks on each sheet. This is meant to ease alignment during exposure.
- `--outline`
Print the contents of the outline layer on each sheet.

`--mirror` Print mirror image.

`--fill-page`
Scale output to make the board fit the page.

`--auto-mirror`
Print mirror image of appropriate layers.

`--ps-color`
Postscript output in color.

`--ps-bloat <num>`
Amount to add to trace/pad/pin edges (1 = 1/100 mil).

`--ps-invert`
Draw objects as white-on-black.

`--media <media-name>`
Size of the media, the postscript is fitted to. The parameter `<media-name>` can be any of the standard names for paper size: 'A0' to 'A10', 'B0' to 'B10', 'Letter', '11x17', 'Ledger', 'Legal', 'Executive', 'A-Size', 'B-size', 'C-Size', 'D-size', 'E-size', 'US-Business_Card', 'Intl-Business_Card'.

`--psfade <num>`
Fade amount for assembly drawings (0.0=missing, 1.0=solid).

`--scale <num>`
Scale value to compensate for printer sizing errors (1.0 = full scale).

`--multi-file`
Produce multiple files, one per page, instead of a single multi page file.

`--xcalib <num>`
Paper width. Used for x-Axis calibration.

`--ycalib <num>`
Paper height. Used for y-Axis calibration.

`--drill-copper`
Draw drill holes in pins / vias, instead of leaving solid copper.

`--show-legend`
Print file name and scale on printout.

6.16 Size-Options

`--via-thickness <num>`
Default diameter of vias in 1/100 mil. Default value is 6000

`--via-drilling-hole <num>`
Default diameter of holes in 1/100 mil. Default value is 2800

`--line-thickness <num>`
Default thickness of new lines in 1/100 mil. Default value is 1000

- `--rat-thickness <num>`
Thickness of rats. Values from 1 to 19 are fixed width in screen pixels. Anything larger means PCB units (i.e. 100 means "1 mil"). Default value is 1000.
- `--keepaway <num>`
Default minimum distance between a track and adjacent copper in 1/100 mil. Default value is 1000.
- `--default-PCB-width <num>`
Default width of the canvas in 1/100 mil. Default value is 600000
- `--default-PCB-height <num>`
Default height of the canvas in 1/100 mil. Default value is 500000
- `--text-scale <num>`
Default text scale. Default value is 100
- `--alignment-distance <num>`
Specifies the distance between the boards outline and alignment targets. Default value is 200
- `--grid <num>`
Initial grid size in 1/100 mil. Default value is 1000
- `--grid-increment-mm <num>`
Metric grid increment in mm. Default value is 0.1
- `--grid-increment-mil <num>`
Imperial grid increment in mil. Default value is 5.0
- `--size-increment-mm <num>`
Metric increment of object size in mm. Default value is 0.2
- `--size-increment-mil <num>`
Imperial increment of object size in mil. Default value is 10.0
- `--line-increment-mm <num>`
Metric increment of line width in mm. Default value is 0.1
- `--line-increment-mil <num>`
Imperial increment of line width in mil. Default value is 5.0
- `--clear-increment-mm <num>`
Metric increment of track clearance in mm. Default value is 0.05
- `--clear-increment-mil <num>`
Imperial increment of track clearance in mil. Default value is 2.0
- `--minimum polygon area <num>`
Minimum polygon area in 1/100 mil.

Index of Actions, Commands and Options

-		--help	41
--basename <string>	40	--invisible-mark-color <string>	37
--action-script <string>	41	--invisible-objects-color <string>	37
--action-string <string>	41	--keepaway <num>	45
--align-marks	43	--layer-color-<n> <string>	37
--alignment-distance <num>	45	--layer-name-1 <string>	40
--all-direction-lines	40	--layer-name-2 <string>	40
--all-layers	39	--layer-name-3 <string>	40
--as-shown	39, 43	--layer-name-4 <string>	40
--auto-mirror	44	--layer-name-5 <string>	40
--background-color <string>	36	--layer-name-6 <string>	40
--backup-interval	41	--layer-name-7 <string>	40
--bg-image <string>	42	--layer-name-8 <string>	40
--black-color <string>	36	--layer-selected-color-<n> <string>	37
--bloat <num>	38	--layer-stack <string>	42
--bomfile <string>	36	--lib-command <string>	38
--clear-increment-mil <num>	45	--lib-command-dir <string>	38
--clear-increment-mm <num>	45	--lib-contents-command <string>	38
--clear-line	39	--lib-name <string>	42
--connected-color <string>	37	--lib-newlib <string>	42
--copper-height <num>	40	--lib-path <string>	42
--copyright	41	--line-increment-mil <num>	45
--cross-color <string>	37	--line-increment-mm <num>	45
--crosshair-color <string>	36	--line-thickness <num>	44
--default-font <string>	42	--listen	42
--default-PCB-height <num>	45	--lprcommand <string>	40
--default-PCB-width <num>	45	--mask-color <string>	37
--dpi	43	--media <media-name>	44
--dpi <num>	40	--menu-file <string>	41
--draw-grid <string>	39	--min-drill <num>	38
--drill-copper	44	--min-ring <num>	38
--drill-helper	43	--min-silk <num>	38
--dump-actions	41	--min-width <num>	38
--element-color <string>	37	--minimum polygon area <num>	45
--element-command <string>	38	--mirror	44
--element-path <string>	41	--monochrome	39, 43
--element-selected-color <string>	37	--multi-file	44
--eps-file <string>	39	--off-limit-color <string>	37
--eps-scale <num>	39	--only-visible	39
--fab-author	42	--only-vivable	43
--file-command <string>	38	--outfile <string>	43
--file-path <string>	42	--outline	43
--fill-page	44	--pcb-menu <string>	42
--font-command <string>	38	--photo-flip-x	43
--font-pat <string>	42	--photo-flip-y	43
--format <string>	43	--photo-mode	43
--full-poly	39	--pin-color <string>	37
--gerberfile	39	--pin-name-color <string>	37
--grid <num>	45	--pin-selected-color <string>	37
--grid-color <string>	37	--pinout-offset-x <num>	39
--grid-increment-mil <num>	45	--pinout-offset-y <num>	39
--grid-increment-mm <num>	45	--pinout-text-offset-x <num>	39
--grid-units-mm	41	--pinout-text-offset-y <num>	39
--groups <string>	41	--png-bloat <num><dim>	43

--print-file <string>.....	38
--ps-bloat <num>.....	44
--ps-color.....	44
--ps-invert.....	44
--psfade <num>.....	44
--psfile <string>.....	43
--rat-color <string>.....	37
--rat-command <string>.....	38
--rat-selected-color <string>.....	37
--rat-thickness <num>.....	45
--reset-after-element.....	42
--ring-bell-finished.....	42
--route-styles <string>.....	41
--save-command <string>.....	38
--save-in-tmp.....	42
--save-last-command.....	42
--scale <num>.....	44
--show-actions.....	41
--show-defaults.....	41
--show-legend.....	44
--show-number.....	40
--shrink <num>.....	38
--size-increment-mil <num>.....	45
--size-increment-mm <num>.....	45
--snap-pin.....	40
--substrate-epsilon <num>.....	41
--substrate-height <num>.....	41
--text-scale <num>.....	45
--unique-names.....	39
--use-alpha.....	43
--verbose.....	39, 41
--version.....	41
--via-color <string>.....	37
--via-drilling-hole <num>.....	44
--via-selected-color <string>.....	37
--via-thickness <num>.....	44
--warn-color <string>.....	37
--x-max.....	43
--xcalib <num>.....	44
--xy-in-mm.....	36
--xy-max.....	43
--xyfile <string>.....	36
--y-max.....	43
--ycalib <num>.....	44
:	
:actionCommand().....	34
:l.....	34
:le.....	34
:m.....	34
:q.....	34
:rn.....	34
:s.....	34
:w[q].....	34

A

AddRats().....	52
ApplyVendor().....	53, 106
Atomic().....	53

B

Bell().....	53
-------------	----

C

ChangeClearSize().....	53
ChangeDrillSize().....	53
ChangeFlag().....	54
ChangeHole().....	54
ChangeName().....	54
ChangeOctagon().....	54
ChangePinName().....	54
ChangeSize().....	54
ChangeSquare().....	54
ClrFlag().....	55
Command().....	55
Connection().....	55

D

DeleteRats().....	55
DisableVendor().....	55, 111
DisperseElements().....	55
Display().....	55
DRC().....	56

E

EditLayerGroups().....	56
EnableVendor().....	56, 115
ExecuteFile().....	56

L

Load().....	57
LoadVendor().....	57
LoadVendorFrom().....	117

M

MarkCrosshair().....	57
Mode().....	57
MovePointer().....	58
MoveToCurrentLayer().....	58

N

New().....	58
------------	----