

**NAME**

pcb2gcode – manual page for pcb2gcode 2.5.0

**DESCRIPTION**

This manual page documents briefly the **pcb2gcode** command.

**pcb2gcode** is a program that takes the files typically produced by PCB (printed circuit board) designing tools, in particular Gerber (RS-274X) and Excellon (an RS-274C variant) files as parsed by **gerbv**, and produces files that can be used to control a CNC (computer numerical control) milling machine. These files are in G code format (RS-274D/NGC) that can be read, for example, by the linuxcnc EMC2 system.

When these files are executed, an engraving bit is used to remove the surface from a copper covered circuit board, thus creating isolated areas. In another step, holes are drilled into the board at the appropriate positions, and it can be cut out in another step. (For each step, a separate output file is created.)

These programs follow the usual GNU command line syntax, with long options starting with two dashes ('-'). A summary of options is included below.

Instead of passing all options on the command line, nearly all options can be stored in a file named *millproject*. There, options are given one per line as *option=value* pairs (without the two dashes), and can be separated by blank lines and commented (everything after a '#' character is treated as a comment). Options that don't take an argument (like **--metric**) are entered as *option=true* or *option=1* there.

Unless configured otherwise, numeric values are in units of inch and inch/minute. When the **--metric** option is given, they are in mm and mm/minute.

pcb2gcode 2.5.0

**command line only options:**

- noconfigfile** [=arg(=1)] (=0) ignore any configuration file
- config** arg (=millproject)  
list of comma-separated config files
- ? [ --help ]**  
produce help message
- V [ --version ]**  
show the current software version

**Generic options (CLI and config files):**

- ignore-warnings** [=arg(=1)] (=0)  
Ignore warnings
- svg** arg  
[DEPRECATED] use **--vectorial**, SVGs will be generated automatically; this option has no effect
- metric** [=arg(=1)] (=0)  
use metric units for parameters. does not affect gcode output
- metricoutput** [=arg(=1)] (=0)  
use metric units for output
- g64** arg  
[DEPRECATED, use tolerance instead] maximum deviation from toolpath, overrides internal calculation
- tolerance** arg  
maximum toolpath tolerance
- nog64** [=arg(=1)] (=0)  
do not set an explicit g64

- output-dir** arg  
output directory
- basename** arg  
prefix for default output file names
- preamble-text** arg  
preamble text file, inserted at the very beginning as a comment.
- preamble** arg  
gcode preamble file, inserted at the very beginning.
- postamble** arg  
gcode postamble file, inserted before M9 and M2.
- no-export** [=arg(=1)] (=0)  
skip the exporting process

#### Drilling options, for making holes in the PCB:

- drill** arg  
Excellon drill file
- milledrill** [=arg(=1)] (=0)  
[DEPRECATED] Use min-milledrill-hole-dia meter=0 instead
- milledrill-diameter** arg  
diameter of the end mill used for drilling with **--milledrill**
- min-milledrill-hole-diameter** arg (=inf)  
minimum hole width or milledrilling. Holes smaller than this are drilled. This implies milledrill
- zdrill** arg  
drilling depth
- zmilledrill** arg  
milledrilling depth
- drill-feed** arg  
drill feed in [i/m] or [mm/m]
- drill-speed** arg  
spindle rpm when drilling
- drill-front** [=arg(=1)]  
[DEPRECATED, use drill-side instead] drill through the front side of board
- drill-side** arg (=auto)  
drill side; valid choices are front, back or auto (default)
- drills-available** arg  
list of drills available
- onedrill** [=arg(=1)] (=0)  
use only one drill bit size
- drill-output** arg (=drill.ngc)  
output file for drilling
- nog91-1** [=arg(=1)] (=0)  
do not explicitly set G91.1 in drill headers
- nog81** [=arg(=1)] (=0)  
replace G81 with G0+G1
- nom6** [=arg(=1)] (=0)  
do not emit M6 on tool changes

**--milldrill-output** arg (=milldrill.ngc)  
output file for milldrilling

#### Milling options, for milling traces into the PCB:

**--front** arg  
front side RS274-X .gbr

**--back** arg  
back side RS274-X .gbr

**--voronoi** [=arg(=1)] (=0)  
generate voronoi regions

**--offset** arg (=0)  
Note: Prefer to use **--mill-diameters** and **--milling-overlap** if you just that's what you mean.  
An optional offset to add to all traces, useful if the bit has a little slop that you want to keep out of the trace.

**--mill-diameters** arg (=0)  
Diameters of mill bits, used in the order that they are provided.

**--milling-overlap** arg (=50 %)  
How much to overlap milling passes, from 0% to 100% or an absolute length

**--isolation-width** arg (=0)  
Minimum isolation width between copper surfaces

**--extra-passes** arg (=0)  
[DEPRECATED] use **--isolation-width** instead. Specify the the number of extra isolation passes, increasing the isolation width half the tool diameter with each pass

**--pre-milling-gcode** arg  
custom gcode inserted before the start of milling each trace (used to activate pump or fan or laser connected to fan)

**--post-milling-gcode** arg  
custom gcode inserted after the end of milling each trace (used to deactivate pump or fan or laser connected to fan)

**--zwork** arg  
milling depth in inches (Z-coordinate while engraving)

**--mill-feed** arg  
feed while isolating in [i/m] or [mm/m]

**--mill-vertfeed** arg  
vertical feed while isolating in [i/m] or [mm/m]

**--mill-infeed** arg  
maximum milling depth; PCB may be cut in multiple passes

**--mill-speed** arg  
spindle rpm when milling

**--mill-feed-direction** arg (=0)  
In which direction should all milling occur

**--invert-gerbers** [=arg(=1)] (=0)  
Invert polarity of front and back gerbers, causing the milling to occur inside the shapes

**--draw-gerber-lines** [=arg(=1)] (=0)  
Draw lines in the gerber file as just lines and not as filled in shapes

**--preserve-thermal-reliefs** [=arg(=1)] (=1)  
generate mill paths for thermal reliefs in voronoi mode

**--front-output** arg (=front.ngc)  
output file for front layer

**--back-output** arg (=back.ngc)  
output file for back layer

#### Outline options, for cutting the PCB out of the FR4:

**--outline** arg  
pcb outline polygon RS274-X .gbr

**--fill-outline** [=arg(=1)] (=1)  
accept a contour instead of a polygon as outline (enabled by default)

**--cutter-diameter** arg  
diameter of the end mill used for cutting out the PCB

**--zcut** arg  
PCB cutting depth in inches

**--cut-feed** arg  
PCB cutting feed in [i/m] or [mm/m]

**--cut-vertfeed** arg  
PCB vertical cutting feed in [i/m] or [mm/m]

**--cut-speed** arg  
spindle rpm when cutting

**--cut-infeed** arg  
maximum cutting depth; PCB may be cut in multiple passes

**--cut-front** [=arg(=1)]  
[DEPRECATED, use cut-side instead] cut from front side.

**--cut-side** arg (=auto)  
cut side; valid choices are front, back or auto (default)

**--bridges** arg (=0)  
add bridges with the given width to the outline cut

**--bridgesnum** arg (=2)  
specify how many bridges should be created

**--zbridges** arg  
bridges height (Z-coordinates while engraving bridges, default to zsafe)

**--outline-output** arg (=outline.ngc)  
output file for outline

#### Optimization options, for faster PCB creation, smaller output files, and different algorithms.:

**--optimise** [=arg(=2.54e-06 m)] (=2.54e-06 m)  
Reduce output file size by up to 40% while accepting a little loss of precision. Larger values reduce file sizes and processing time even further.  
Set to 0 to disable.

**--eulerian-paths** [=arg(=1)] (=1)  
Don't mill the same path twice if milling loops overlap. This can save up to 50% of milling time.  
Enabled by default.

**--vectorial** [=arg(=1)] (=1)  
enable or disable the vectorial rendering engine

**--tsp-2opt** [=arg(=1)] (=1)  
use TSP 2OPT to find a faster toolpath (but slows down gcode generation)

- path-finding-limit** arg (=1)  
Use path finding for up to this many steps in the search (more is slower but makes a faster gcode path)
- g0-vertical-speed** arg (=0.0211667 m s<sup>-1</sup>)  
speed of vertical G0 movements, for use in path-finding
- g0-horizontal-speed** arg (=0.0423333 m s<sup>-1</sup>)  
speed of horizontal G0 movements, for use in path-finding
- backtrack** arg (=inf)  
allow retracing a milled path if it's faster than retract-move-lower. For example, set to 5in/s if you are willing to remill 5 inches of trace in order to save 1 second of milling time.

**Autolevelling options, for generating gcode to automatically probe the board and adjust milling depth to the actual board height:**

- al-front** [=arg(=1)] (=0)  
enable the z autoleveller for the front layer
- al-back** [=arg(=1)] (=0)  
enable the z autoleveller for the back layer
- software** arg  
choose the destination software (useful only with the autoleveller). Supported programs are linux-cnc, mach3, mach4 and custom
- al-x** arg  
max x distance between probes
- al-y** arg  
max y distance between probes
- al-probefeed** arg  
speed during the probing
- al-probe-on** arg (=MSG, Attach the probe tool)@M0 ( Temporary machine stop. )  
execute this commands to enable the probe tool (default is M0)
- al-probe-off** arg (=MSG, Detach the probe tool)@M0 ( Temporary machine stop. )  
execute this commands to disable the probe tool (default is M0)
- al-probecode** arg (=G31)  
custom probe code (default is G31)
- al-probevar** arg (=2002)  
number of the variable where the result of the probing is saved (default is 2002)
- al-setzzero** arg (=G92 Z0)  
gcode for setting the actual position as zero (default is G92 Z0)

**Alignment options, useful for aligning the milling on opposite sides of the PCB:**

- x-offset** arg (=0)  
offset the origin in the x-axis by this length
- y-offset** arg (=0)  
offset the origin in the y-axis by this length
- zero-start** [=arg(=1)] (=0)  
set the starting point of the project at (0,0)
- mirror-absolute** [=arg(=1)] (=1)  
[DEPRECATED, must always be true] mirror back side along absolute zero instead of board center

**--mirror-axis** arg (=0)

For two-sided boards, the PCB needs to be flipped along the axis x=VALUE

**--mirror-yaxis** arg (=0)

For two-sided boards, the PCB needs to be flipped along the y axis instead

**CNC options, common to all the milling, drilling, and cutting:**

**--zsafe** arg

safety height (Z-coordinate during rapid moves)

**--spinup-time** arg (=0.001 s)

time required to the spindle to reach the correct speed

**--spindown-time** arg

time required to the spindle to return to 0 rpm

**--zchange** arg

tool changing height

**--zchange-absolute** [=arg(=1)] (=0)

use zchange as a machine coordinates height (G53)

**--tile-x** arg (=1)

number of tiling columns. Default value is 1

**--tile-y** arg (=1)

number of tiling rows. Default value is 1

Git commit: Boost: 108300 Gerbv: 2.10.0 Geos: Not installed