

---

# FreePCB post-processing tool

fpconvert can produce g-code and HPGL output for CNC engraver or plotter

## Table of Contents

fpconvert.sh .....	2
generate.sh .....	3
fpconvert.conf .....	4

---

## Name

fpconvert — A FreePCB post-processing tool

## Description

```
fpconvert.sh { [-m] | [-d] | [-r] | [-u] } [-f [(1)format ] ] [-l [(2)layer ] ] [-o file ] [-e depth ] [-x] [-c file.conf ] [-h] { file.fpc }
```

(1) [hpgl] | [gcode]

(2) [top] | [bottom]

**fpconvert** has been designed to create a G-code output for CNC engraver.

---

## Name

generate.sh — A fpconvert batch

## Description

```
generate.sh { file.fpc } [hpgl]
```

Assumption: The `fpc` file is prepared in separate directory and `fpconvert.conf` for this file is inside the same directory. Batch **generate.sh** creates following g-code files:

- bottom drilling file: `.bot.dri.ngc`
- top drilling file: `.top.dri.ngc`
- bottom isolation milling file: `.bot.ngc`
- top isolation milling file: `.top.ngc`
- bottom solder mask UV laser hardening file: `.bot.sm.ngc`
- top solder mask UV laser hardening file: `.top.sm.ngc`

The batch has been written for PCB setup simplification. User must initiate `fpconvert.conf` file in chosen directory. In addition, there is another config file, called `fpconvert.sm.conf` in the same directory, which is generated automatically by this batch. This file is used for `*.sm.ngc` files generation. If an argument `[hpgl]` is used, output is in `hpgl` format and extensions of generated files are changed to `.plt`.

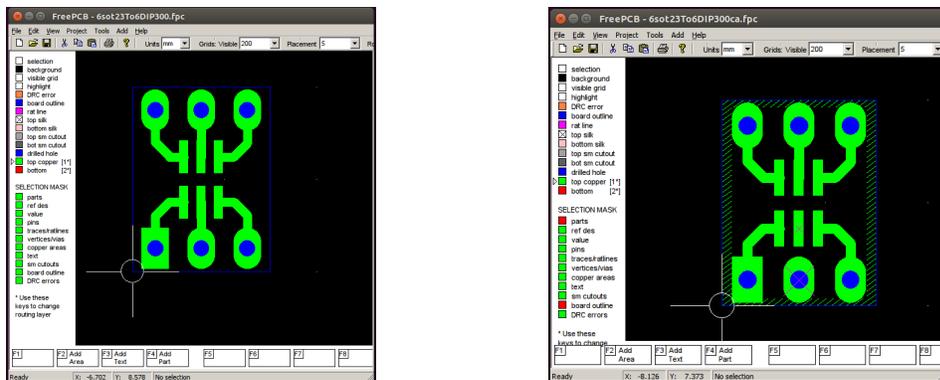
# Name

fpconvert.conf — The configuration file for **fpconvert** .

# Description

Variables are stored in separated rows. Comment character is #. Commented default values are stored in initial fpconvert.conf file. The file fpconvert.conf is the default config file and can be changed using [-c] option. Units are supported for options specified by length units. Number and unit must not be separated by space. Valid values are NM (default), UM, MM, CM, DM, MIL, IN, FT, YD. I.e. 1.5MM means one and half millimeters, 1MIL means one mil.

**Figure 1. Consider following two examples of FreePCB design. Let's use these examples as main explanation of following parameters.**



# Global options

These options are useful for all operations.

## N\_VERT\_CIRCLE\_APPROX

Circles are approximated using n-tagons. This value specifies the number of n-tagon's vertices. Default value is 25.

## X\_SHIFT

Specifies the distance in given units, which is added to all x coordinates before rotation

## Y\_SHIFT

Specifies the distance in given units, which is added to all y coordinates before rotation

## INVERT\_X\_AXIS

Boolean type. If true is chosen, all output x points are multiplied by -1 before rotation. Setting this value to true is possible also using [-x] option.

## INVERT\_Y\_AXIS

Boolean type. If true is chosen, all output y points are multiplied by -1 before rotation

## WORKING\_AREA\_LEFT\_BOTTOM\_POINT

Coordinates of working area left bottom point in given units. NEGATIVE\_WORKING\_AREA\_WIDTH\_HALF, 0MM is set as default, where

`NEGATIVE_WORKING_AREA_WIDTH_HALF` means  $-0.5 * \text{WORKING\_AREA\_WIDTH}$ . This parameter can be used for final working area positioning as well as `X_SHIFT` and `Y_SHIFT` parameters.

#### `WORKING_AREA_WIDTH`

Output working area width (in x coordinate) specification. Main recommendation is several millimeters less than milled PCB width.

#### `WORKING_AREA_HEIGHT`

Output working area height (in y coordinate) specification. Main recommendation is several millimeters less than milled PCB height.

#### `USE_WORKING_AREA_DIMENSIONS_FOR_AXIS_INVERTING`

Boolean type. If `true` is chosen, inverted axes are shifted by values specified in `WORKING_AREA_WIDTH`, or `WORKING_AREA_HEIGHT`, respectively. Default value is `false`.

#### `PROCESS_PADS_ONLY`

Boolean type. If `true` is chosen, only pads will remain from all nets. This is useful for solder mask creation using DLI method.

#### `THERMAL_CONNECTIONS_WIDTH`

Specifies the pads thermal connection width inside copper areas.

#### `CLOCKWISE_ROTATION`

Boolean type. If `true` is chosen, the x and y coordinates of all points are swapped. Using this parameter is not recommended. Note: Main swap operation is processed after axis shifting operations. Thus meaning of `X_SHIFT` and `Y_SHIFT` is swapped if `CLOCKWISE_ROTATION` is set to `true`. The same rule is valid for `INVERT_X_AXIS` and `INVERT_Y_AXIS` parameters.

## Width adjustment

#### `ADDED_CONN_WIDTH_HALF`

Connections width modification in nanometers - i.e. value 150UM means that all connections will become for 150 micrometers wider.

#### `ADDED_PAD_RADIUS`

Pads radius modification.

#### `ADDED_VIA_RADIUS`

Vias radius modification.

#### `ADDED_TEXT_WIDTH`

Non-inverted texts width modification.

#### `ADDED_HOLE_DIAMETER_WIDTH`

Holes diameter modification.

## Z axis adjustment

These options makes sense only for *gcode* output format.

#### `Z_START_POSITION`

The initial position of Z axis.

#### `Z_MOVING_POSITION`

The initial position of Z axis. This position is also used for tool changing.

**Z\_ZERO\_POSITION**

This value sets the Z axis position, where the *G00* (fast moving) command became to the *G01* (work moving) command.

## Milling options

**Z\_MILL\_DEPTH**

Mill depth specification.

**FEED\_RATE\_MILLING**

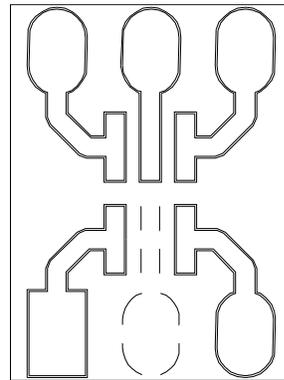
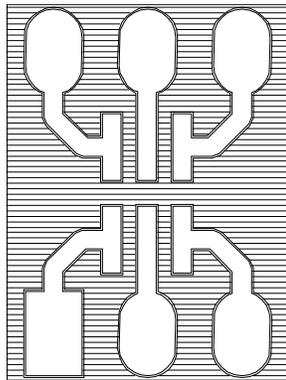
Feed rate milling specification in millimeters per minute. Default value is 150.

**MILLING\_TOOL\_WIDTH**

Tool width used for milling.

**N\_CONTOURS**

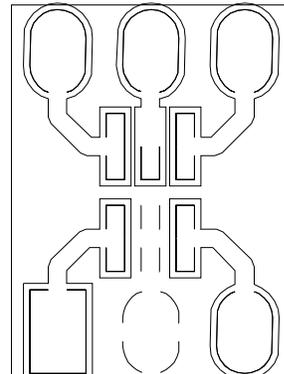
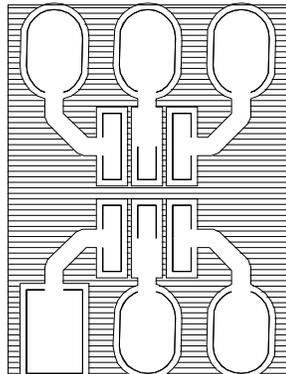
This option is a way, how to increase the net isolation width. See the **GAP\_WIDTH\_BETWEEN\_CONTOURS** option how to set the space between contours. The processing order is largest contour first. Default value is one contour per net.

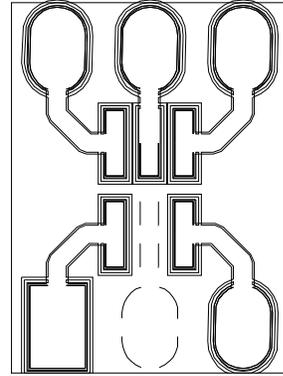
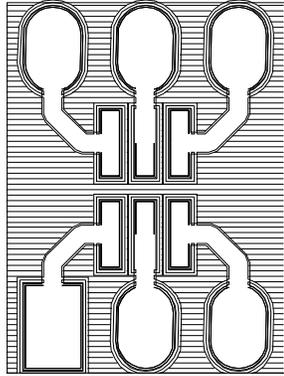
**GAP\_WIDTH\_BETWEEN\_CONTOURS**

If option **N\_CONTOURS** is used, this option sets the space between net contours.

**ADDED\_PAD\_ISOLATION\_WIDTH**

This is the way how to increase isolation gap around pads only. The isolation is created at the expense of isolation pad dimensions. It can be combined with options **N\_CONTOURS** and **GAP\_WIDTH\_BETWEEN\_CONTOURS**.





### MILL\_ORDER

**fpconvert** generates output code by steps. These steps can be specified. All steps are set as the default value in following order: BOARD NETS UNCONNECTED\_PADS THERMAL\_PADS THERMAL\_VIAS TEXTS RESIDUAL\_COPPER

BOARD	Mills the board outline.
NETS	Mills all nets including connected pads and vias.
UNCONNECTED_PADS	Mills pads, which do not have assigned nets.
THERMAL_PADS	Mills thermal pads, inside copper areas.
THERMAL_VIAS	Mills thermal vias, inside copper areas.
TEXTS	Mills texts. FreePCB uses only one font type - <i>hershey</i>
RESIDUAL_COPPER	This step is most complicated. All steps are reproduced in rasterization mode. See the section called "Rasterization options" for mode details. If this step is used, it should be placed as final.

## Drilling options

These options makes sense only for *gcode* output format.

### Z\_TOOL\_CHANGE\_POSITION

Z axis position when the drill is prepared for manual changing.

### X\_TOOL\_CHANGE\_POSITION

x axis position when the drill is prepared for manual changing. Useful when a clamp is used for PCB fixation.

### Z\_DRILL\_DEPTH

Drill depth specification.

### FEED\_RATE\_DRILLING

Feed rate drawing specification in millimeters per minute. Default value is 150.

### DWELL\_PERIOD\_DRILLING\_IN\_MS

Dwell duration, when the drill is in bottom position - in milliseconds. Designed for cleaner holes.

**N\_HOLE\_DRILL\_REPEAT**

N - times drill hole repeat. The default value is 1. This should be a way how to obtain cleaner holes.

**N\_HOLE\_DRILL\_REPEAT\_ADDED\_DIAMETER**

Adds this value to select proper dill for next drilling iteration. Default value is 0.1MM.

**N\_HOLE\_DRILL\_REPEAT\_MAX\_DIAMETER**

Sets N\_HOLE\_DRILL\_REPEAT to 1 (stops repeat) for holes with larger diameter than this value. Default value is 1MM.

**DRILL\_HOLES\_IN\_BOTTOM\_CORNERS**

Boolean type. If `true` is set, left and right bottom corners inside working area (see parameters `WORKING_AREA_WIDTH`, `WORKING_AREA_HEIGHT` and `USE_WORKING_AREA_DIMENSIONS_FOR_AXIS_INVERTING`) are drilled by bit size, which is given by `HOLE_IN_BOTTOM_CORNER_DIAMETER`. This can be useful for double sided board manipulation. Using these holes is possible to match top layer with X axis mirrored bottom layer. Default value is `false`.

**HOLE\_IN\_BOTTOM\_CORNER\_DIAMETER**

Sets a drill bit size for `DRILL_HOLES_IN_BOTTOM_CORNERS`. Default value is 1.2MM.

**DRILL\_HOLE\_IN\_ORIGIN**

Boolean type. If `true` is set, The hole with diameter specified by `HOLE_IN_ORIGIN_DIAMETER` is drilled. This is useful, when CNC axes homing is needed more than once - i.e. a clamp is used for drilling holes, but for milling is board held by double sided adhesive tape. Default value is `false`.

**HOLE\_IN\_ORIGIN\_DIAMETER**

Sets a drill bit size for `DRILL_HOLE_IN_ORIGIN`. Default value is 0.5MM.

**DRILL\_BIT\_DIAMETERS**

Ascending sorted list of drills to be used for the drilling process. Values

0.5MM 0.6MM 0.7MM 0.8MM 0.9MM 1MM 1.1MM 1.2MM

are set as default.

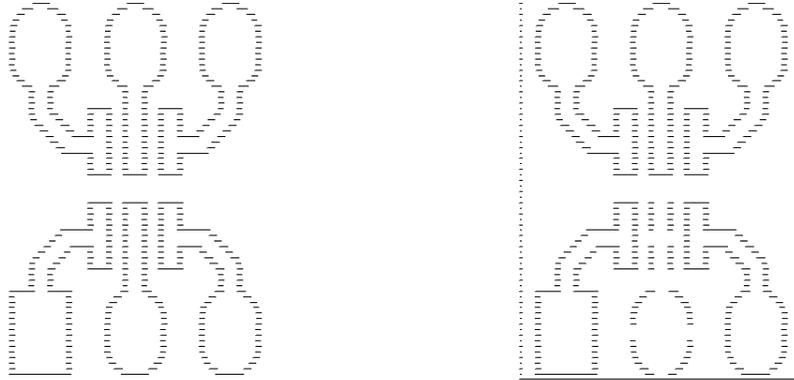
**DRILL\_BIT\_TOLERANCE**

If diameter of processed hole is currently chosen diameter, specified in `DRILL_BIT_DIAMETERS` plus or minus `DRILL_BIT_TOLERANCE` value, this diameter is chosen as a final tool. Default value is 0.5MM.

## Rasterization options

**USE\_RASTERIZATION**

Boolean type. If `true` is chosen, the output will be generated line-by-line style. The distance of lines is specified using `RASTERIZATION_DPI` parameter. For example, this is useful for solder mask hardening.



**RASTERIZATION\_DPI**

DPI specification for rasterization process. Default value is 1200 .

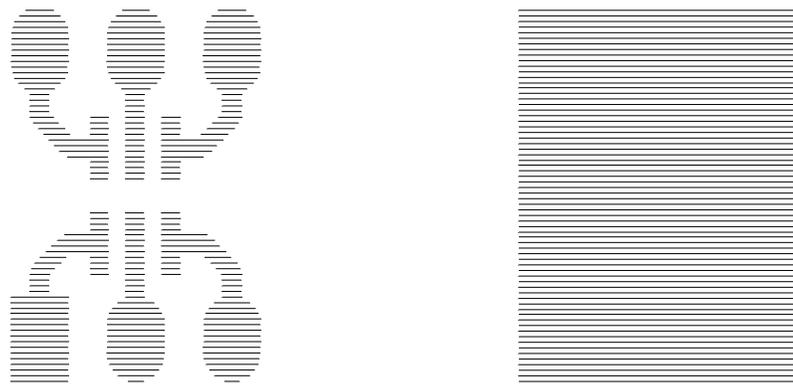
**RASTERIZATION\_INVERT**

Boolean type. If `true` is chosen, the meaning of output lines is inverted. This is useful for negative resist or solder mask hardening in DLI method. Images below show what will happen if parameters `RASTERIZATION_DPI` , `RASTERIZATION_INVERT` and `RASTERIZATION_FILL_POLYGONS` are set to `true` .



**RASTERIZATION\_FILL\_POLYGONS**

Boolean type. If `true` is chosen, all polygons are filled. This is useful for all milling operations.



**RASTERIZATION\_TOOL\_WIDTH**

A width of physical processing tool. Default value is 150UM. If RASTERIZATION\_INVERT is chosen, the meaning of output lines is inverted. This is useful for negative resist or solder mask hardening in DLI method. Note: For virtual drawing into bitmap is used parameter MILLING\_TOOL\_WIDTH .

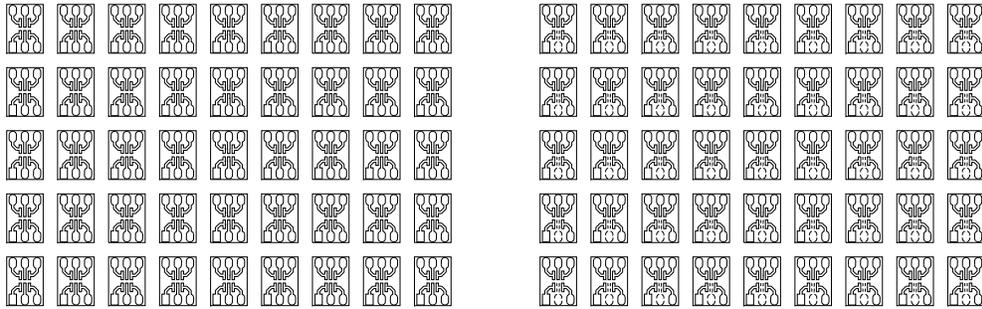
**RASTERIZATION\_COLOR\_INVERT\_SWAP\_DRAW\_PRIORITY\_THRESHOLD****RASTERIZATION\_FILL\_POLYGONS\_SWAP\_DRAW\_PRIORITY\_THRESHOLD**

Nets have priorities with default value 100. Lower value means higher priority. Copper area net type has priority equals 10.

## Panelization options

**USE\_PANELIZATION**

Boolean type. If `true` is chosen, the process will be repeat for each panel shifted by board dimension plus values specified by parameter `GAP_X_BETWEEN_PANELS` or `GAP_Y_BETWEEN_PANELS` . The number of panels is auto-calculated using parameters `WORKING_AREA_WIDTH` and `WORKING_AREA_HEIGHT` . Note: Do not use panelization option together with `CLOCKWISE_ROTATION` option. Temporarily the `RESIDUAL_COPPER` value in `MILL_ORDER` is not supported. Thus for panelization this step must not be used.

**GAP\_X\_BETWEEN\_PANELS**

X coordinate distance between panels. Default value is 3MM .

**GAP\_Y\_BETWEEN\_PANELS**

Y coordinate distance between panels. Default value is 3MM .

## Drawing options

### DLI options

**S\_LASER\_INTENSITY\_COMMANDS**

Boolean type. If `true` is chosen, all processing g-code working commands are changed to `M3` and `M5` . This can be used for laser driver connection to spindle PWM output in CNC machine.