fpcconvert tutorial

Pavel Štemberk

fpcconvert tutorial

Pavel Štemberk

Table of Contents

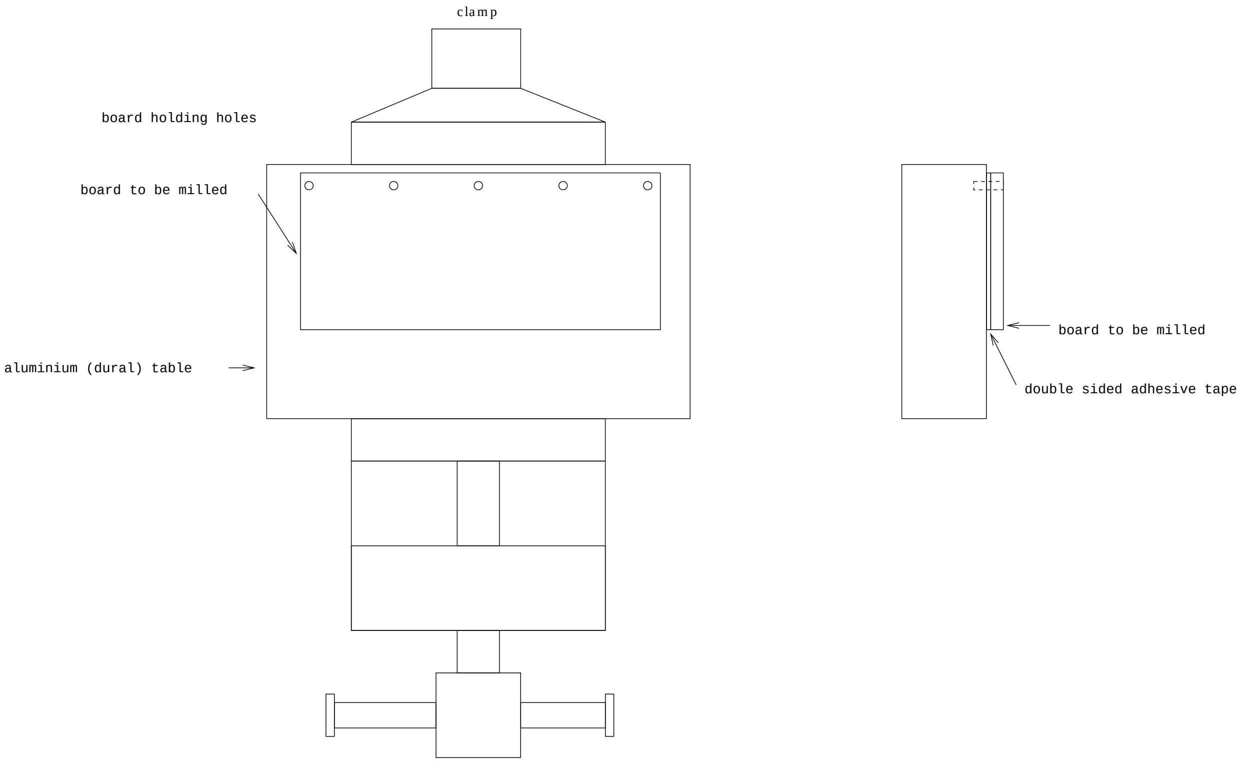
List of Figures

Chapter 1. Equipment

Hardware

The main concept of PCB manufacturing process is shown in the figure below. The board is fixed using double sided adhesive tape on the aluminum table. This table is flatten using appropriate milling cutter first. This is really important preparation setup for milling the PCB.

Figure 1.1. Aluminum table held by clamp



Chapter 2. Non-panelized PCB

Figure 2.1. Stand-alone board

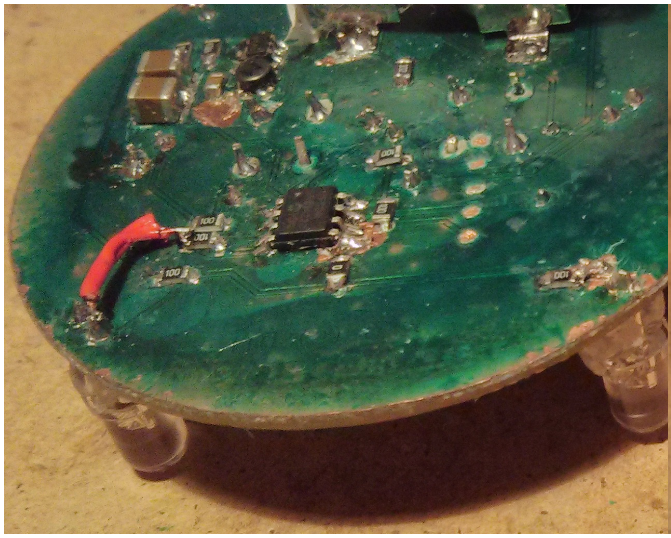
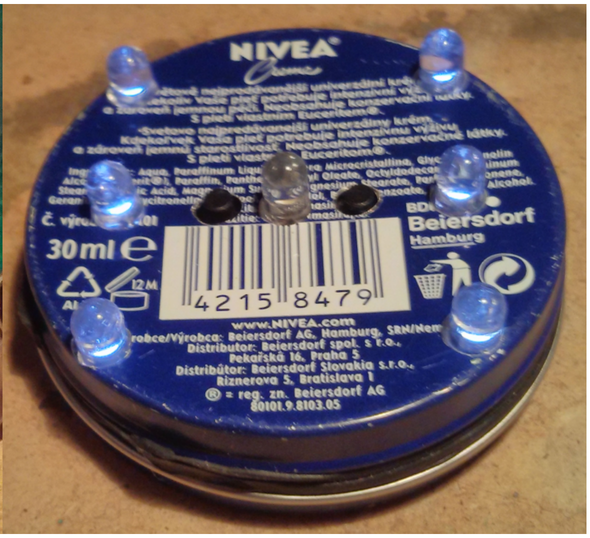


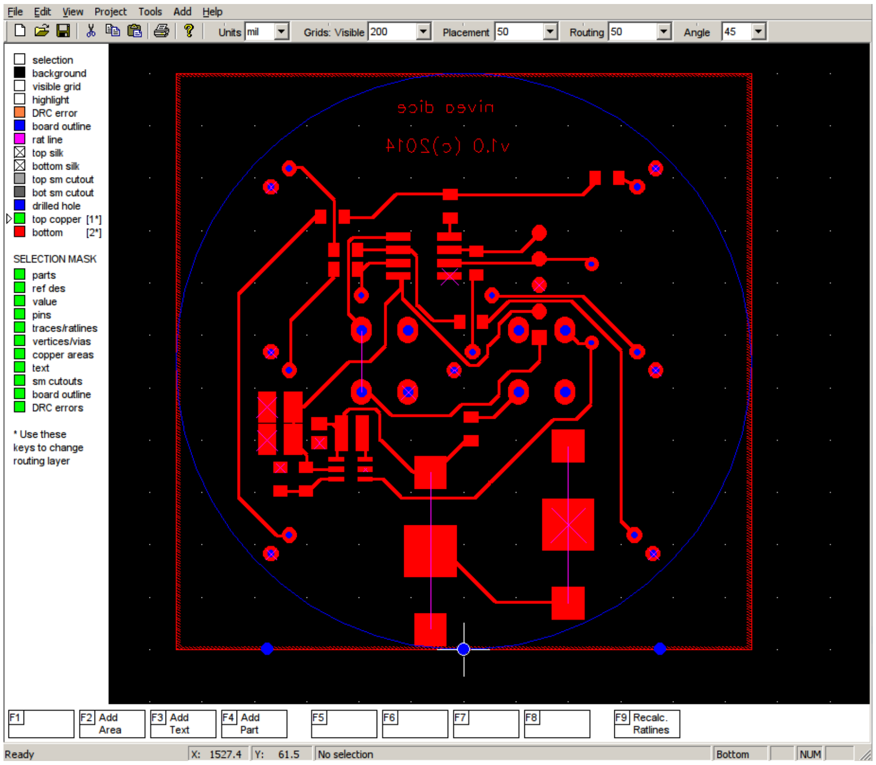
Figure 2.2. Finished dice



Design customization

First, check our design. Board must be centered by Y-axis as is shown in Figure 2.3, “Project in FreePCB”. It is quite convenient to prepare holding holes plus one hole in origin point. In this case there are placed at bottom.

Figure 2.3. Project in FreePCB



Units mil
Grids: Visible 200
Placement 50
Routing 50
Angle 45
X: 1527.4 Y: 61.5

Customize configuration file

Main consideration is that the one FreePCB project is placed into stand-alone directory with no other projects. Then we can copy the fpccovert.conf.default file in fpcconvert directory to fpcconvert.conf in our project directory or create a new file called fpcconvert.conf and write following lines:

```
N_VERT_CIRCLE_APPROX   = 28
ADDED_CONN_WIDTH_HALF  = 75UM
ADDED_PAD_RADIUS       = 75UM
ADDED_VIA_RADIUS       = 75UM
Z_START_POSITION       = 10MM
Z_MOVING_POSITION      = 2MM
Z_MILL_DEPTH           = -0.25MM
X_TOOL_CHANGE_POSITION = 90MM
Z_DRILL_DEPTH          = -4.3MM
FEED_RATE_DRILLING     = 40
```

N_VERT_CIRCLE_APPROX is increased to 28 due to smoother board outline, which has circle shape. Pads, connections and vias are increased by 75 micrometers. Z_MILL_DEPTH is most important parameter and should be always negative. X_TOOL_CHANGE_POSITION is set to 90 millimeters to allow drill bit change, because it could not be possible when spindle is above on the clamp. The PCB board is held directly by clamp without aluminum table, so there is a lot of space to drill. This is the reason why Z_DRILL_DEPTH is set more than twice of board thickness. FEED_RATE_DRILLING is slowed to 40 millimeters per minute.

Generate g-code file for drilling

We will drill into the bottom side. So the parameter for x axis mirroring [-x] is used.

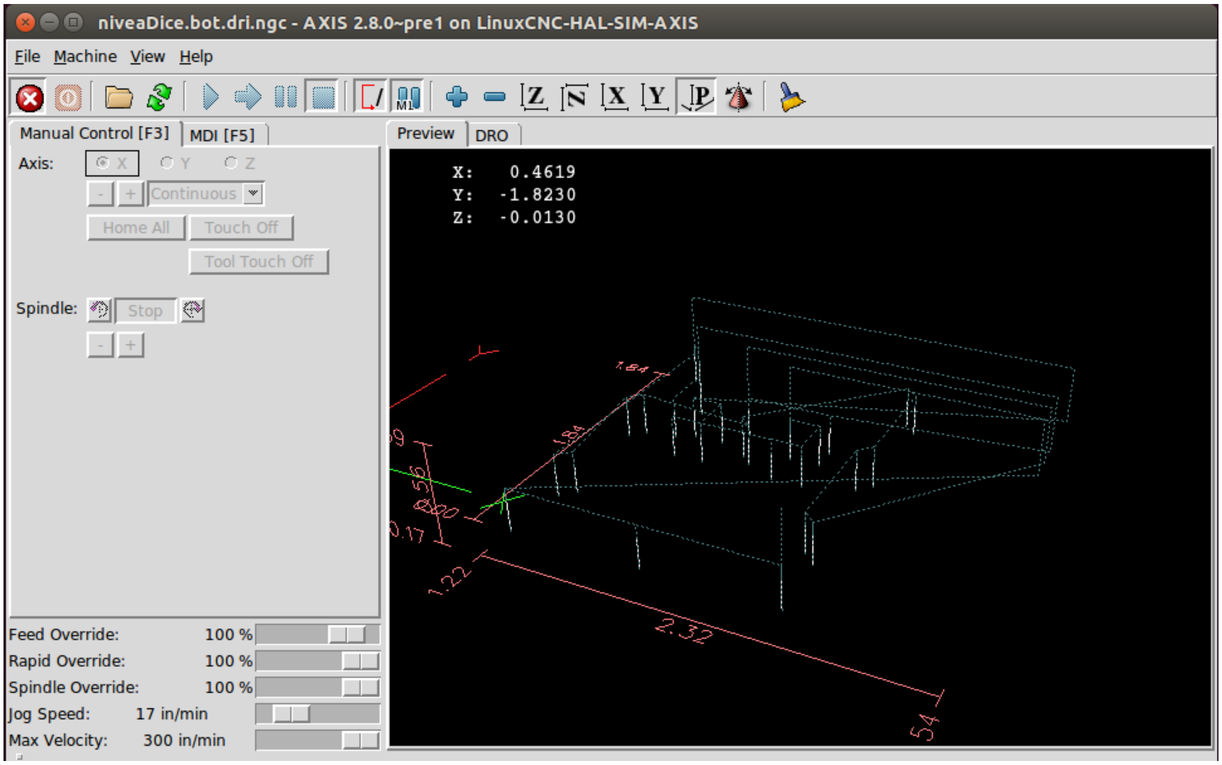
```
./fpcconvert.sh -c examples/niveaDice/fpcconvert.conf -xr examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.bot.dri.ngc -lbottom
```

Following text is returned:

```
setting config file to examples/niveaDice/fpcconvert.conf

Warning: zero thickness for connection (4 -> 3), segment 1, net N000022
Warning: zero thickness for connection (2 -> 1), segment 1, net N000003
Warning: zero thickness for connection (3 -> 1), segment 1, net N000004
so library loaded, turbo GPC mode will be used
```

Warnings reported point to unconnected nets in the design, see figure Figure 2.3, “Project in FreePCB”. This is on purpose in this case.

Figure 2.4. LinuxCNC - drilling holes



```
X:    0.4619
Y:   -1.8230
Z:   -0.0130
```

Feed Override: 100 %
Rapid Override: 100 %
Spindle Override: 100 %
Jog Speed: 17 in/min
Max Velocity: 300 in/min

Generate top and bottom g-code files for milling

If we will mill into the bottom side, we must set the parameter for x axis mirroring. Following command generates appropriate g-code file for bottom (x-axis mirrored) side:

```
./fpcconvert.sh -c examples/niveaDice/fpcconvert.conf -xm examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.bot.ngc -lbottom
```

If we have a double sided PCB, we should also generate the g-code file for top layer:

```
./fpcconvert.sh -c examples/niveaDice/fpcconvert.conf -m examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.top.ngc -ltop
```

Note: In this case we assumed the project as one-sided and there no copper area created on top layer. This leads to wrong result shown in Figure 2.6, “Top layer”. A way how to fix it, is to remove RESIDUAL_COPPER from MILL_ORDER:

```
MILL_ORDER = BOARD NETS UNCONNECTED_PADS THERMAL_PADS THERMAL_VIAS TEXTS
```

Figure 2.5. Bottom layer

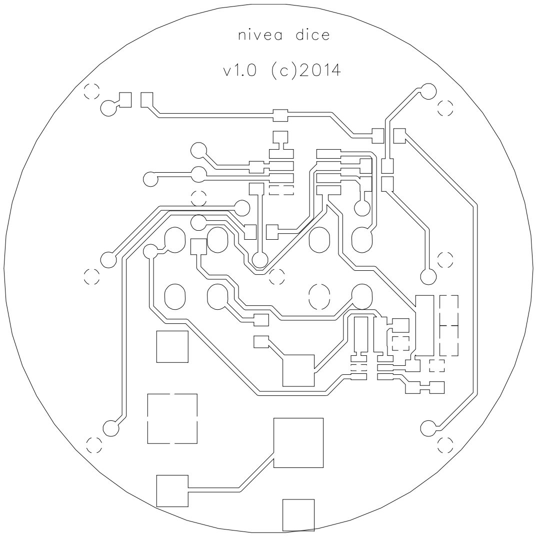
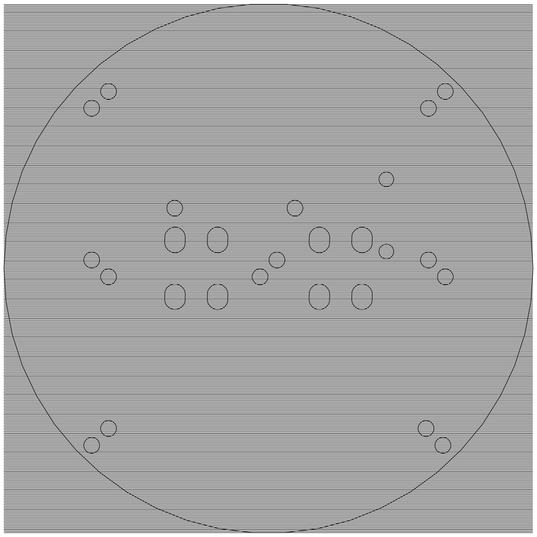


Figure 2.6. Top layer



Generate top and bottom g-code files for solder mask hardening

There are 2 methods how to apply solder mask on our PCB. First consists in use of solder mask dry resist film. Second - we can use a cheap special solution primary used for PCB repairs. Both methods should be successfully applicable to our project. Another modification of config file is necessary for this step. Copy existing config file fpcconvert.config to fpcconvert.sm.config in project directory. Next, change/add following lines:

```
ADDED_PAD_RADIUS           = 150UM
FEED_RATE_MILLING          = 200
USE_RASTERIZATION          = true
RASTERIZATION_INVERT       = true
RASTERIZATION_FILL_POLYGONS = true
RASTERIZATION_TOOL_WIDTH    = 250UM
S_LASER_INTENSITY_COMMANDS  = true
PROCESS_PADS_ONLY          = true
```

ADDED_PAD_RADIUS increases radius of all used pads for 150 micrometers. Feed rate is increased to 200 millimeters per minute. Setting RASTERIZATION_TOOL_WIDTH to 250 micrometers gives 4 lines per millimeter. PROCESS_PADS_ONLY tells **fpcconvert** that no connections between pad will be drawn. To solder mask creation, **fpcconvert** uses component's pads instead of solder mask layers as they are defined in FreePCB. The command syntax is similar to process described in the section called "Generate top and bottom g-code files for milling". Bottom side:

```
./fpcconvert.sh -c examples/niveaDice/fpcconvert.sm.conf -xm examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.bot.sm.ngc -lbottom
```

Top side:

```
./fpcconvert.sh -c examples/niveaDice/fpcconvert.sm.conf -m examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.top.sm.ngc -ltop
```

Figure 2.7. Bottom layer solder mask

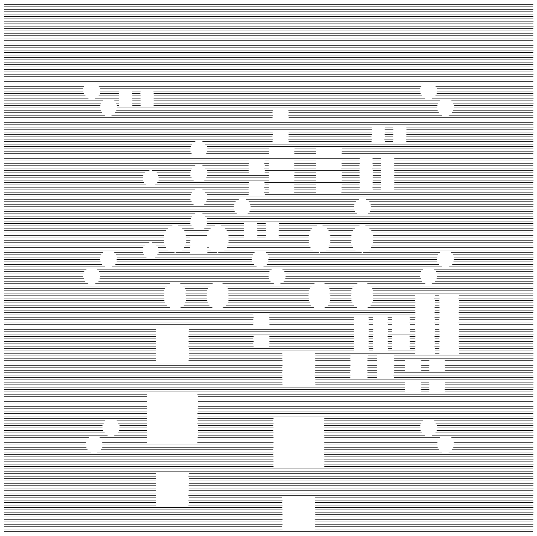
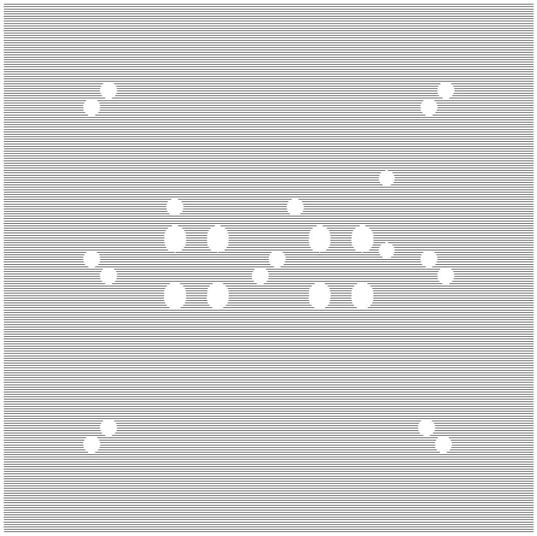


Figure 2.8. Top layer solder mask



Using batch generate.sh

To make steps described in the section called "Generate g-code file for drilling", the section called "Generate top and bottom g-code files for milling", the section called "Generate top and bottom g-code files for solder mask hardening" and the section called "Using batch generate.sh" at once, there is a batch command **generate.sh**.

```
./generate.sh examples/niveaDice/niveaDice.fpc
```

Output:

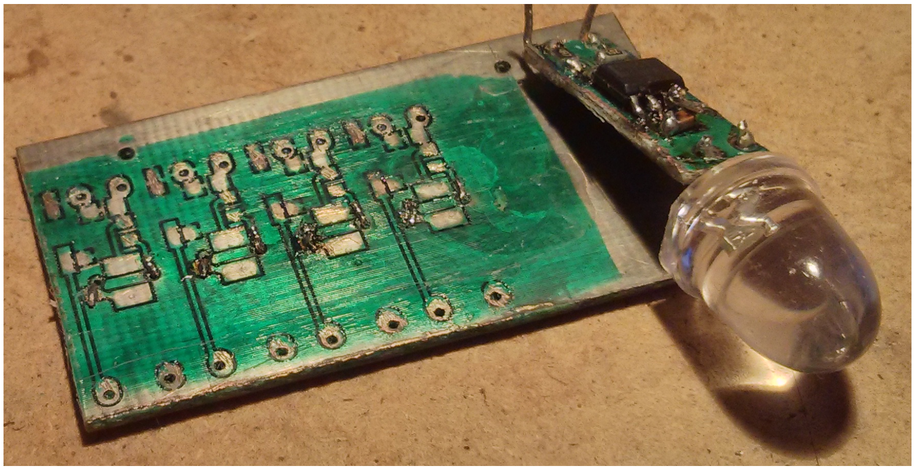
```
./fpcconvert.sh -c examples/niveaDice/fpcconvert.conf -xr examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.bot.dri.ngc -lbottom
setting config file to examples/niveaDice/fpcconvert.conf

Warning: zero thickness for connection (4 -> 3), segment 1, net N000022
Warning: zero thickness for connection (2 -> 1), segment 1, net N000003
Warning: zero thickness for connection (3 -> 1), segment 1, net N000004
so library loaded, turbo GPC mode will be used
./fpcconvert.sh -c examples/niveaDice/fpcconvert.conf -r examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.top.dri.ngc -lbottom
setting config file to examples/niveaDice/fpcconvert.conf

Warning: zero thickness for connection (4 -> 3), segment 1, net N000022
Warning: zero thickness for connection (2 -> 1), segment 1, net N000003
Warning: zero thickness for connection (3 -> 1), segment 1, net N000004
so library loaded, turbo GPC mode will be used
./fpcconvert.sh -c examples/niveaDice/fpcconvert.conf -xm examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.bot.ngc -lbottom
setting config file to examples/niveaDice/fpcconvert.conf

Warning: zero thickness for connection (4 -> 3), segment 1, net N000022
Warning: zero thickness for connection (2 -> 1), segment 1, net N000003
Warning: zero thickness for connection (3 -> 1), segment 1, net N000004
so library loaded, turbo GPC mode will be used
./fpcconvert.sh -c examples/niveaDice/fpcconvert.conf -m examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.top.ngc -ltop
setting config file to examples/niveaDice/fpcconvert.conf

Warning: zero thickness for connection (4 -> 3), segment 1, net N000022
Warning: zero thickness for connection (2 -> 1), segment 1, net N000003
Warning: zero thickness for connection (3 -> 1), segment 1, net N000004
so library loaded, turbo GPC mode will be used
creating fpcconvert.sm.conf in examples/niveaDice
./fpcconvert.sh -c examples/niveaDice/fpcconvert.sm.conf -xm examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.bot.sm.ngc -lbottom
setting config file to examples/niveaDice/fpcconvert.sm.conf

Warning: zero thickness for connection (4 -> 3), segment 1, net N000022
Warning: zero thickness for connection (2 -> 1), segment 1, net N000003
Warning: zero thickness for connection (3 -> 1), segment 1, net N000004
so library loaded, turbo GPC mode will be used
./fpcconvert.sh -c examples/niveaDice/fpcconvert.sm.conf -m examples/niveaDice/niveaDice.fpc -fgcode -o examples/niveaDice/niveaDice.top.sm.ngc -ltop
setting config file to examples/niveaDice/fpcconvert.sm.conf

Warning: zero thickness for connection (4 -> 3), segment 1, net N000022
Warning: zero thickness for connection (2 -> 1), segment 1, net N000003
Warning: zero thickness for connection (3 -> 1), segment 1, net N000004
so library loaded, turbo GPC mode will be used
```

Note: This batch command is written in bash, so it has been tested in Linux only.

Chapter 3. Panelized PCB

We use this method when we need more than one PCB board.

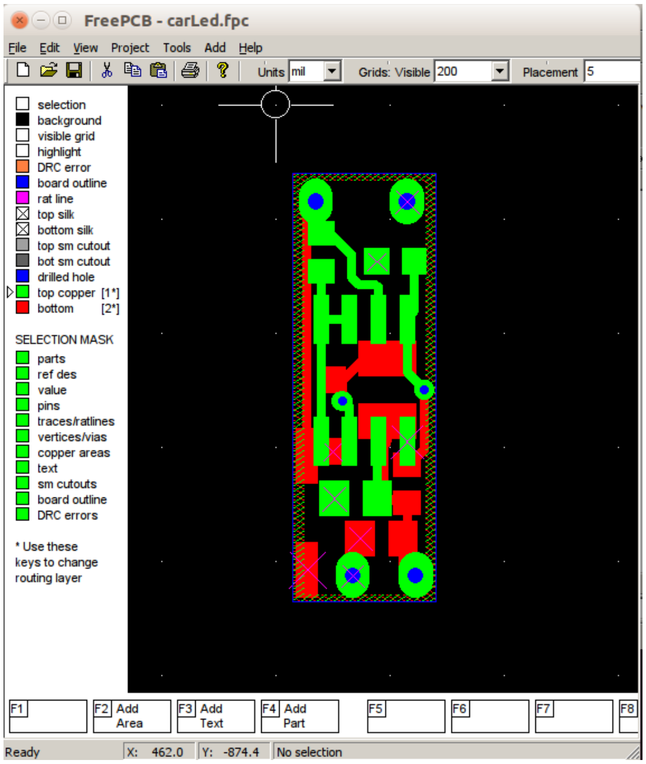
Figure 3.1. Panelized PCBs and finished car parking bulb



Design customization

It's important if the board is placed top, or bottom (this case) of x-axis. This leads to proper y-axis expansion during panelization. There is not y-axis centering request as it was in previous chapter. Also auxiliary board holding holes will be created automatically.

Figure 3.2. Project in FreePCB



Units mil
Grids: Visible 200
Placement 5

Ready X: 462.0 Y: -874.4 No selection

Customize configuration file

Prepare fpcconvert.conf file similar way how is described in the section called “Customize configuration file” Edit or add following lines:

```
ADDED_CONN_WIDTH_HALF = 75000NM
ADDED_PAD_RADIUS = 75000NM
ADDED_VIA_RADIUS = 75000NM
Z_START_POSITION = 10MM
Z_MOVING_POSITION = 2MM
Z_MILL_DEPTH = -0.33MM
X_TOOL_CHANGE_POSITION = 90MM
Z_DRILL_DEPTH = -3.3MM
FEED_RATE_DRILLING = 50
FEED_RATE_CUTTING = 30
Y_SHIFT_IN_MM = 2
USE_WORKING_AREA_DIMENSIONS_FOR_AXIS_INVERTING = true
USE_PANELIZATION = true
GAP_X_BETWEEN_PANELS = 1MM
GAP_Y_BETWEEN_PANELS = 1MM
WORKING_AREA_WIDTH = 1.5IN
WORKING_AREA_HEIGHT = 25MM
DRILL_HOLES_IN_BOTTOM_CORNERS = true
DRILL_HOLE_IN_ORIGIN = true
MILL_ORDER = NETS UNCONNECTED_PADS THERMAL_PADS THERMAL_VIAS TEXTS
DRILL_BIT_DIAMETERS = 0.4MM 0.5MM 0.6MM 0.7MM 0.8MM 0.9MM 1MM 1.1MM 1.2MM
```

Let's describe most important configuration parameters: Setting USE_PANELIZATION to true turns on the panelization process. Algorithm tries to fit a maximum number of our boards into the place described by WORKING_AREA_WIDTH and WORKING_AREA_HEIGHT using board dimensions, GAP_X_BETWEEN_PANELS and GAP_Y_BETWEEN_PANELS.

Setting USE_WORKING_AREA_DIMENSIONS_FOR_AXIS_INVERTING to true is necessary for panelization process, setting parameters DRILL_HOLES_IN_BOTTOM_CORNERS and

DRILL_HOLE_IN_ORIGIN automatically drill auxiliary holes, which has been mentioned in previous chapter. MILL_ORDER has been depleted for BOARD and RESIDUAL_COPPER here.

Note: Real size of used board must be larger than is specified by WORKING_AREA_WIDTH and WORKING_AREA_HEIGHT. These parameters tells **fpcconvert** operating dimensions mainly used for mirroring.

Remaining steps

The rest is identical as is described in previous chapter.

Figure 3.3. LinuxCNC - drilling holes

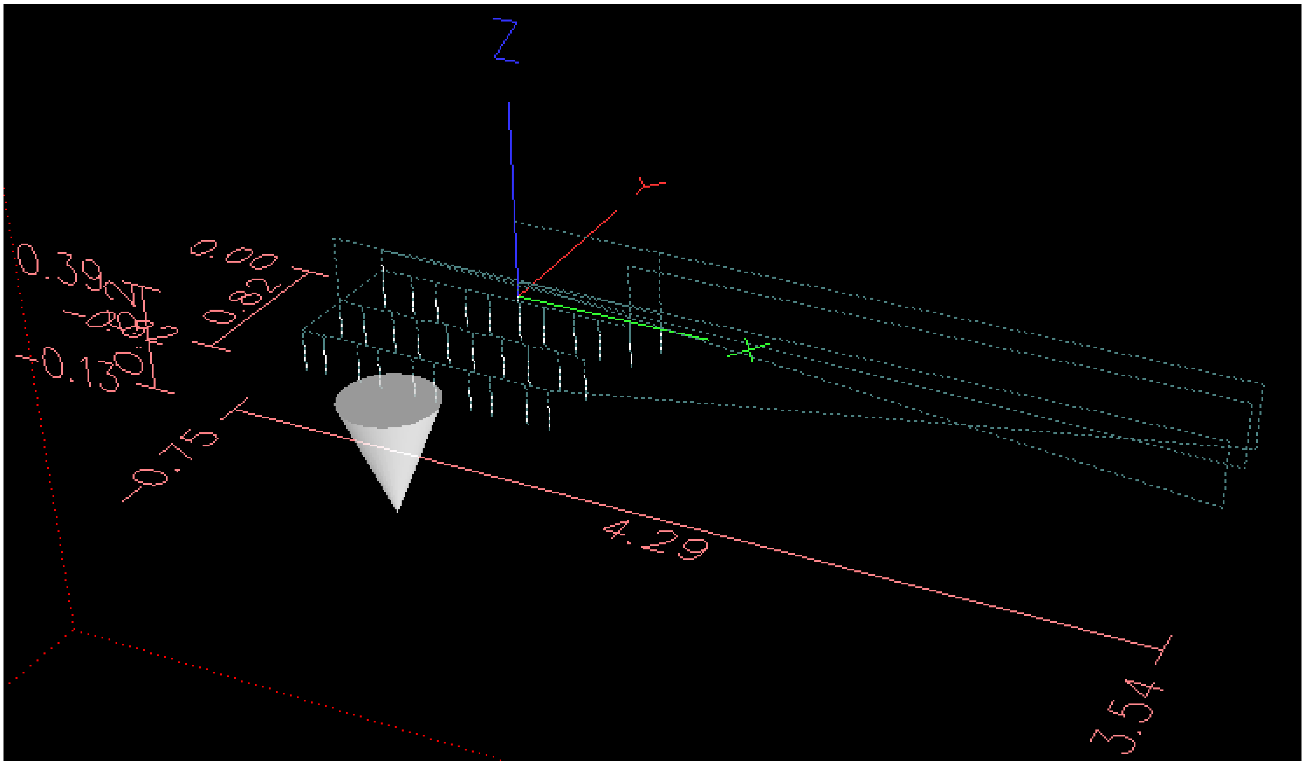
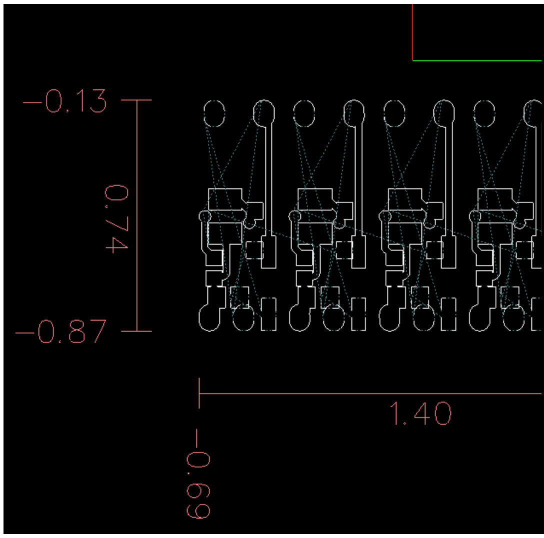


Figure 3.4. Bottom layer solder mask

Figure 3.5. Top layer solder mask



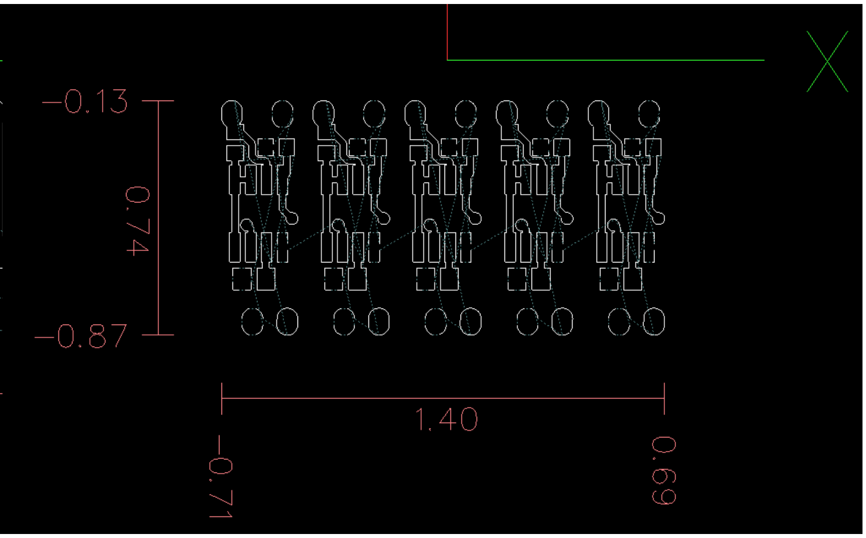
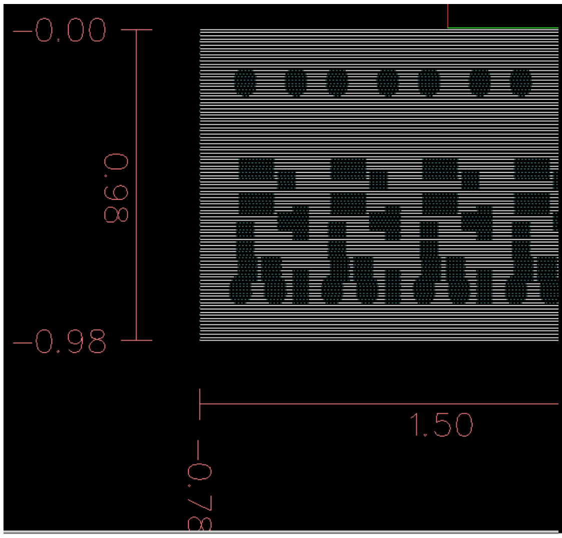
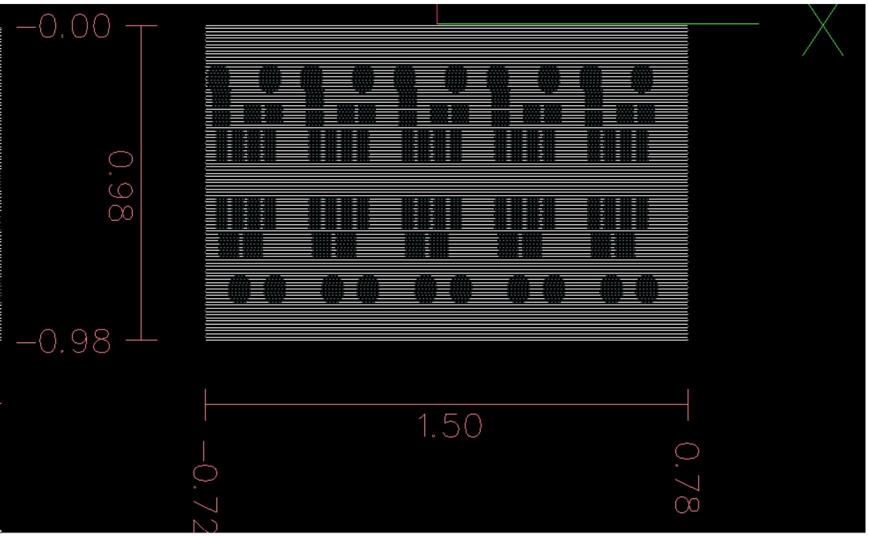


Figure 3.6. Bottom layer solder mask

Figure 3.7. Top layer solder mask





Chapter 4. Another project

Figure 4.1. Bottom

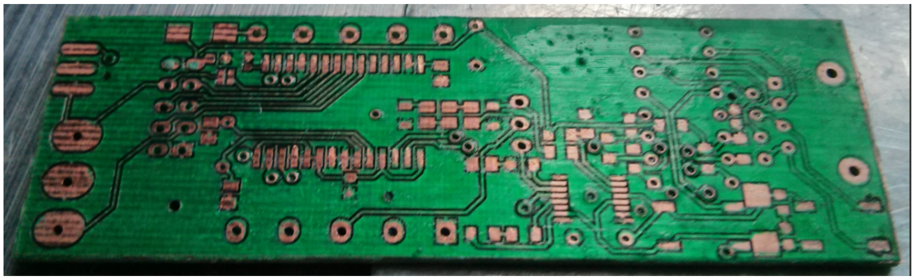


Figure 4.2. Top