PCB with CNC
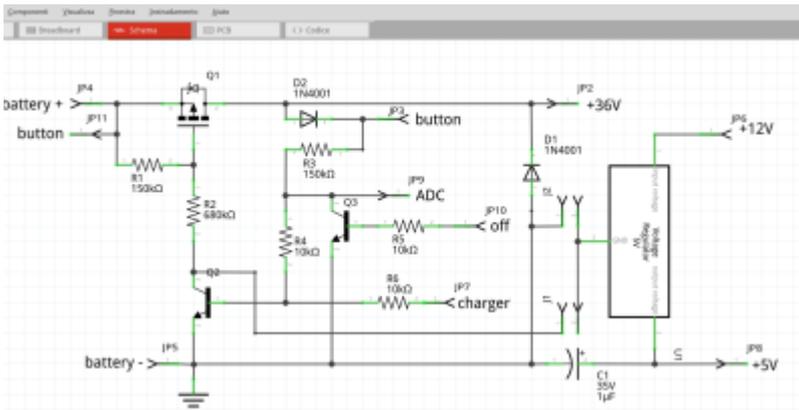
- reference
- fritzing
- online jscut guide
- online ncviewer
- online drill to gcode converter
- TODO online carbide3d

CNC for PCB:

- 22000RPM
- V-Bit cutter 30 degree 0.2mm
- first pass 0.04mm
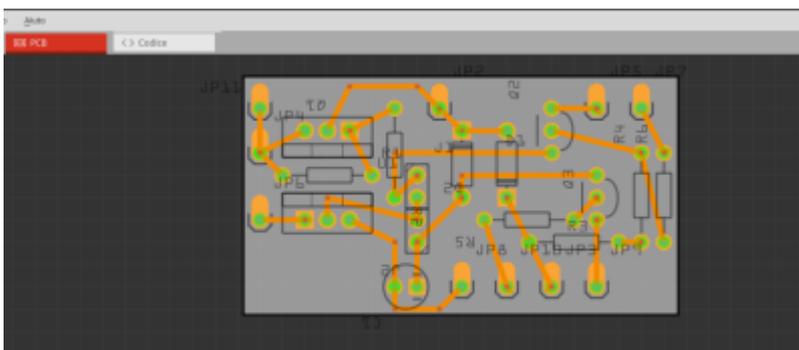- second pass 0.07mm with more feed
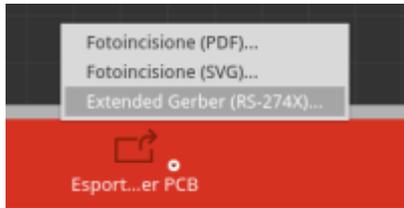
fritzing

Make electronic schema with fritzing

Make pcb routing:

- routing → DRC settings → 32mil large
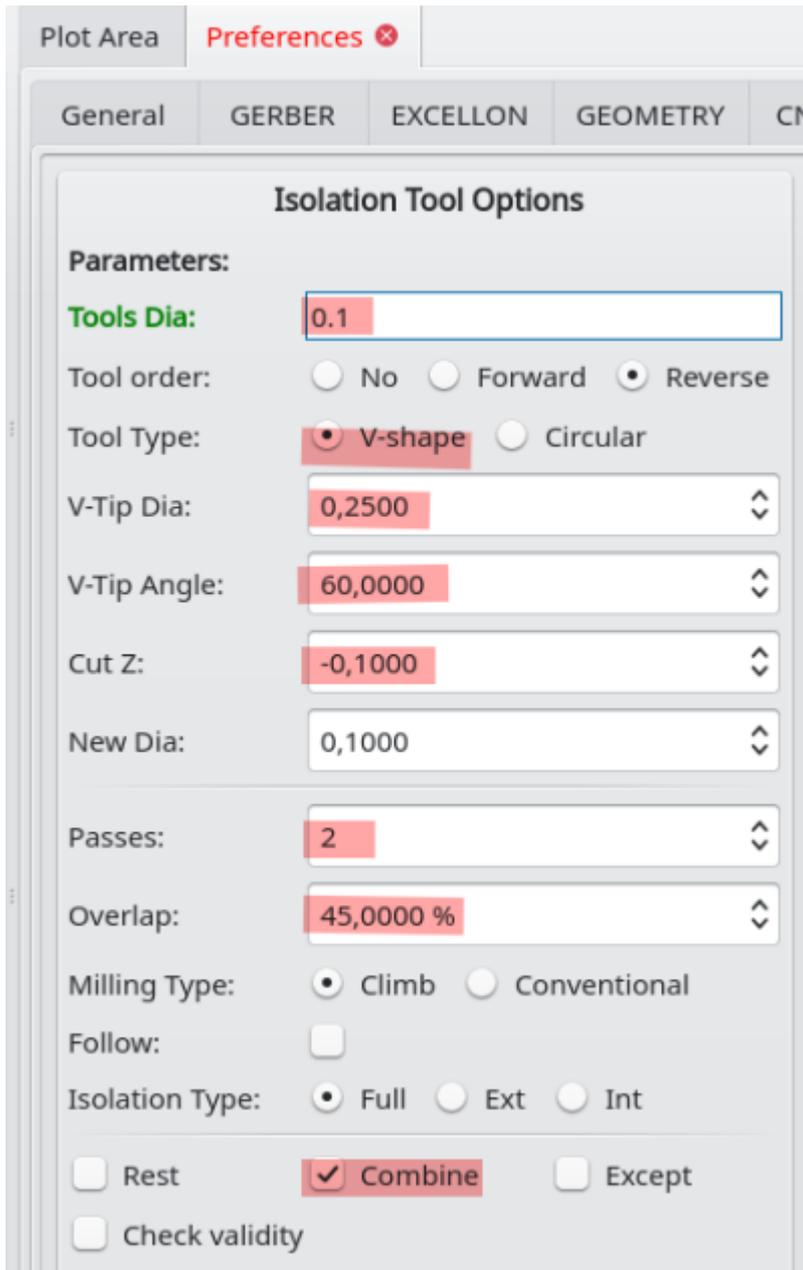- routing → DRC (check design at the end)

export gerber files

flatcam

setup flatcam

Cutout Tool Options

Parameters:

Tool Diameter: 2,0000

Cut Z: -2,0000

☑ Multi-Depth: 0,2000

Kind: ● Single ○ Panel

Margin: 0,1000

Gap size: 4,0000

openfiles.webm

mirror.webm

bottom.webm

top.webm

drill.webm

open gerber files (top and bottom if present) and exellon drill file
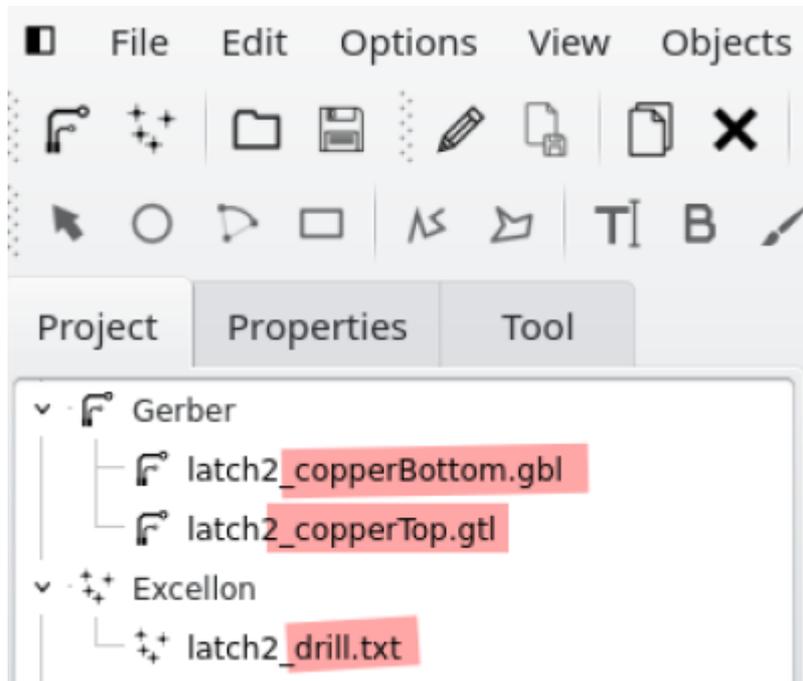
select copper top and
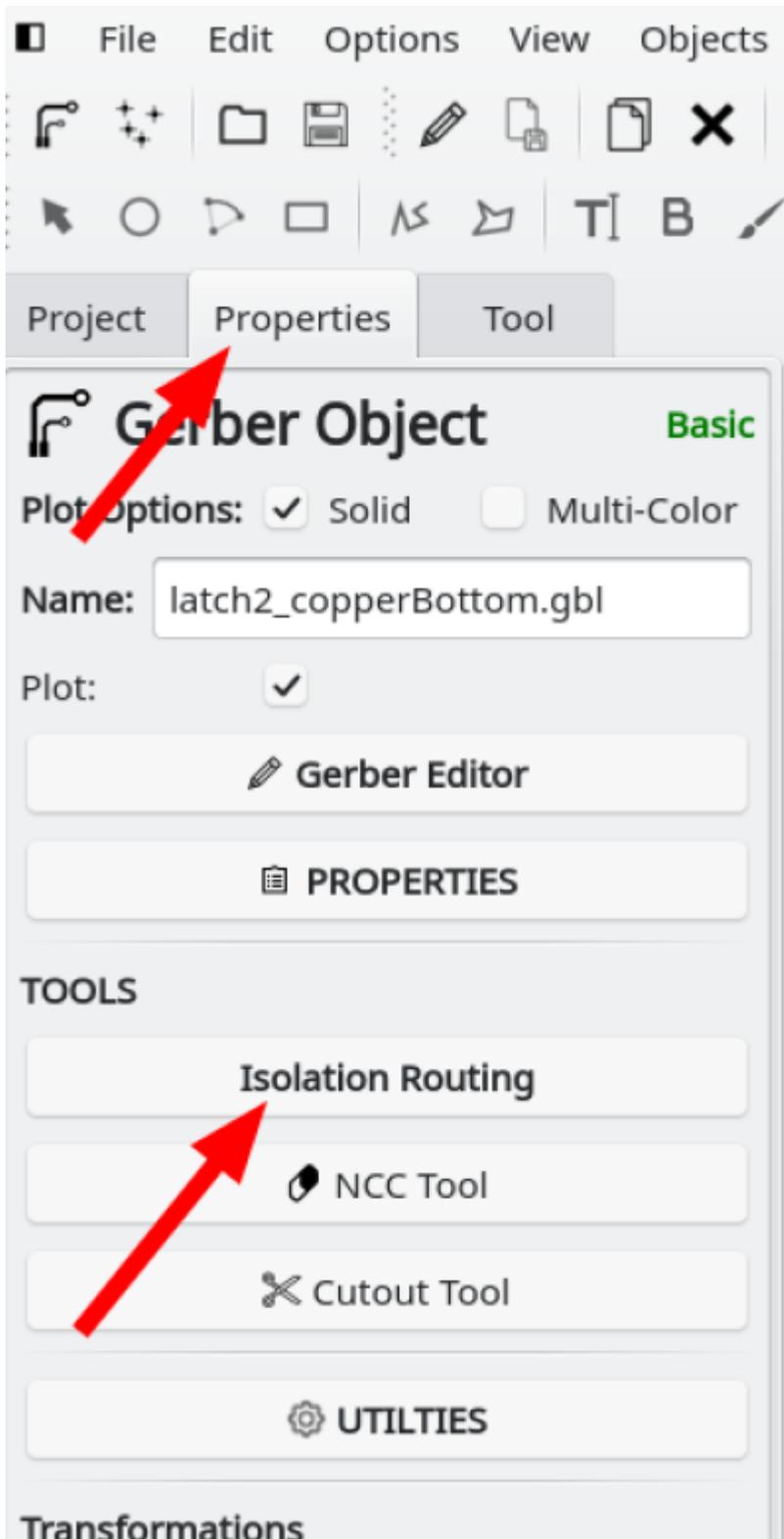
delete tool from tool table 'pick from db' and 'generate geometry'
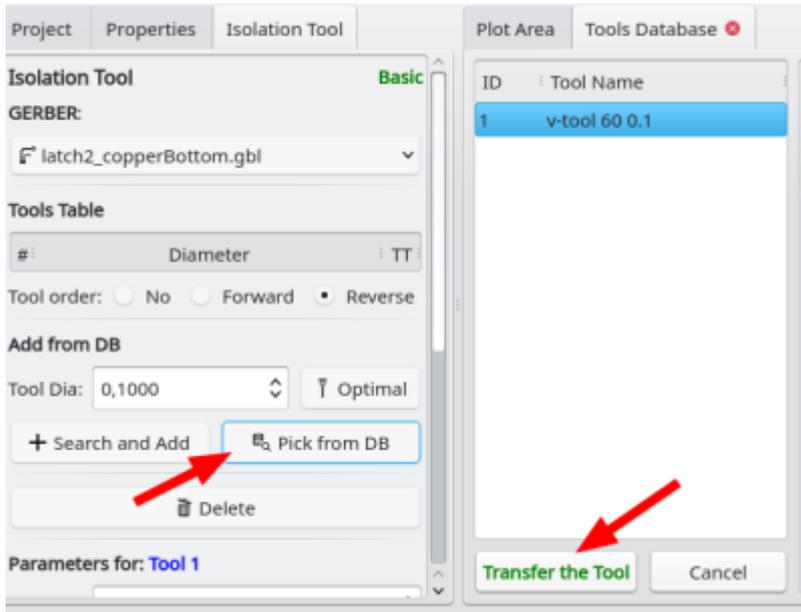
generate cnc job object

| Project | Properties | Isolation Tool |

Parameters for: Tool 1

V-Tip Dia:	0,1000
V-Tip Angle:	60,0000
Cut Z:	-0,0800
☐ Multi-Depth:	0,8000
Travel Z:	1,0000
Feedrate X-Y:	60,0000
Feedrate Z:	60,0000
Spindle speed:	6000
☐ Dwell:	1,0000

✂ Apply parameters to all tools

Common Parameters

☐ Tool change Z:	15,0000
End move Z:	15,0000
End move X,Y:	None
Preprocessor:	default

☐ Add exclusion areas

☐ Add Polish

.𝚈. Generate CNCJob object

save gcode file
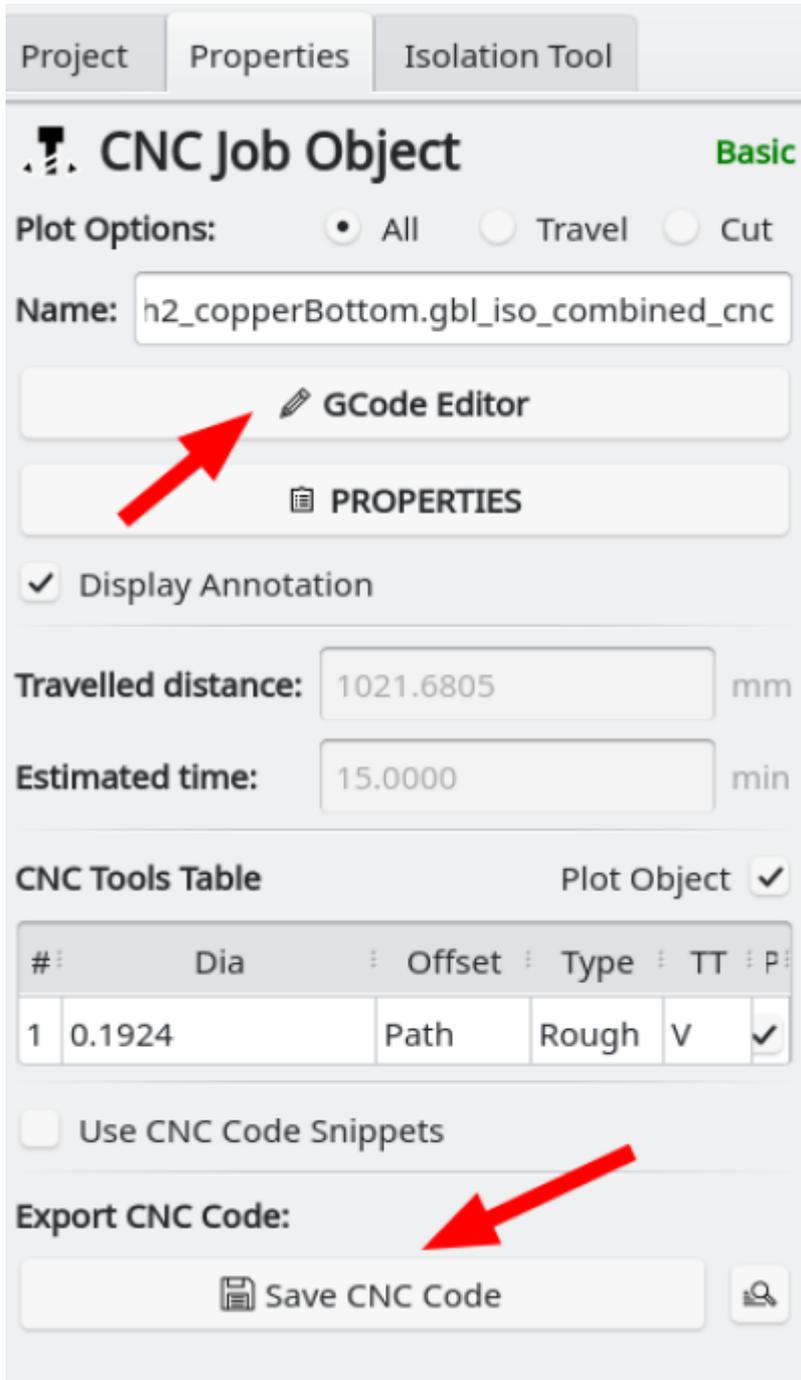
now select *copperBottom.gbl file and make same job to produce gcode for bottom layer. But bottom layer has to be mirrored before geometry object creation.

2-Sided PCB

Source Object:

Type: ● Gerber   ○ Geometry   ○ Excellon

⌐ latch2_copperBottom.gbl ⌄

Bounds Values:

X min:	0,0000
Y min:	0,0000
X max:	0,0000
Y max:	0,0000

| Centroid: | Center point coordinates |

Calculate Bounds Values

Mirror Operation:

Axis: ○ X

         ● Y (1)   (2)

Reference: ○ Point   ○ Box   ● Hole Snap

(3.1013, 30.3047)

Excellon: ⁘ latch2_drill.txt ⌄

Pick hole (3)

⇄ **Mirror**
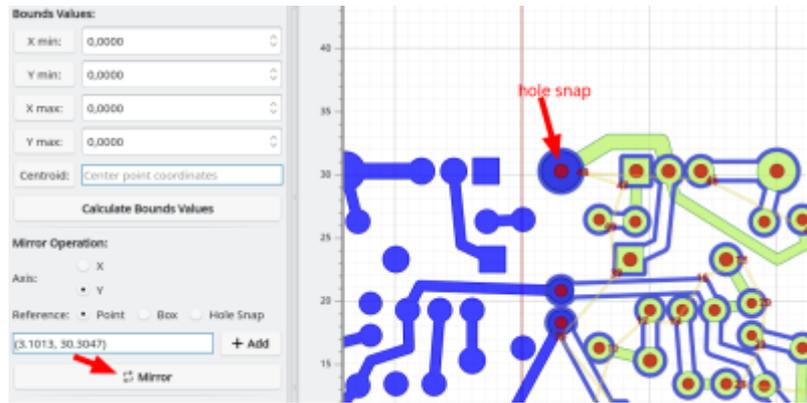
drill job

| Project | Properties | Tool |

⁺₊⁺ Excellon Object **Basic**

Plot Options: ✓ Solid   ☐ Multi-Color

Name: latch2_drill.txt

✎ **Excellon Editor**

▤ **PROPERTIES**

Tools Table   Plot ✓

#	Diameter	Drills	Slots
...	0.7620	2	
...	0.8890	15	
...	0.9000	3	
...	0.9652	3	
...	1.0000	3	
...	1.0160	2	
...	1.0668	11	
...	1.1000	2	
Total Drills		**41**	
Total Slots			**0**

TOOLS

📖 **Drilling Tool**

🛠 Milling Tool

⚙ **UTILTIES**

deprecated

convert gerber to svg

Upload <name>*copperBottom.gbl* to https://tracespace.io, *select layers, download zip and extract* <*name*>copperBottom.gbl

Now with inkscape we need to process the complex SVG with multiples path, objects, layers, group, etc. to convert it into a single path simple SVG. You need to open your SVG into Inkscape (Open-Source, cross-platform), and perform the following series of commands:

- CTRL+A (Select all),
- CTRL+U (Ungroup),
- CTRL+ALT+C (Convert stroke to path),
- CTRL+A (Select all),
- CTRL+U (Union) and
- CTRL+SHIFT+R (Fit page to content).
- CTRL+S save

svg to gcode

Open [jscut](#):

- load SVG <name>_copperBottom.gbl
- make all mm (link)
- zero lower left (link)
- select all objets, drills and path
- create operation
    - outside, 0.1mm

- generate
- save gcode

edit gcode to set splindle speed to 8000 rpm and turn on it clockwise (after G90)

```
; after first G1
M3 S5000
```

carbide3d

set splindle speed to 8000 rpm and turn on it clockwise (after G90)

```
G00 Z1.000
M3 S10000
```

substitute G01 F400 with G01 F40

substiture G00 Z3.000 with G00 Z1.000

deprecated

- export your PCB as .svg files by clicking on "Export for PCB" on the bottom. Click on the small arrow on the Export button and select "Etchable (SVG)". You will get a bunch of svg's exported in your selected directory but we will only use two of them:
  - yourfilename*etchcopperbottommirror.svg * yourfilenameetch*mask*bottom*mirror.svg