

PCB with CNC

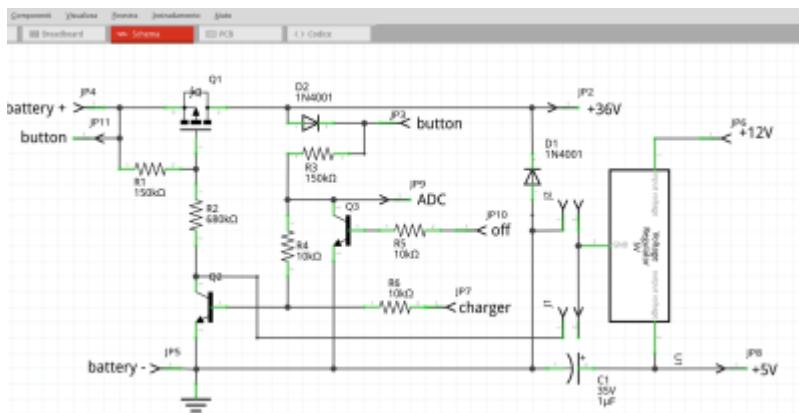
- reference
- fritzing
- online [jscut guide](#)
- online [ncviewer](#)
- online [drill to gcode converter](#)
- TODO online [carbide3d](#)

CNC for PCB:

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

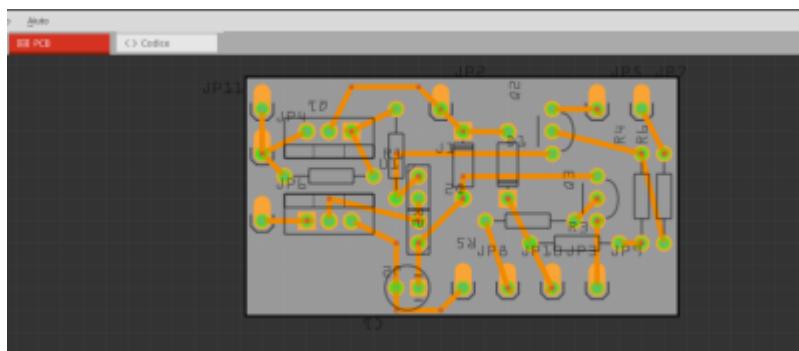
fritzing

Make electronic schema with fritzing

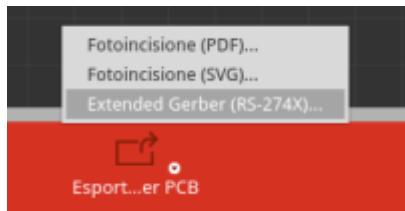


Make pcb routing:

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

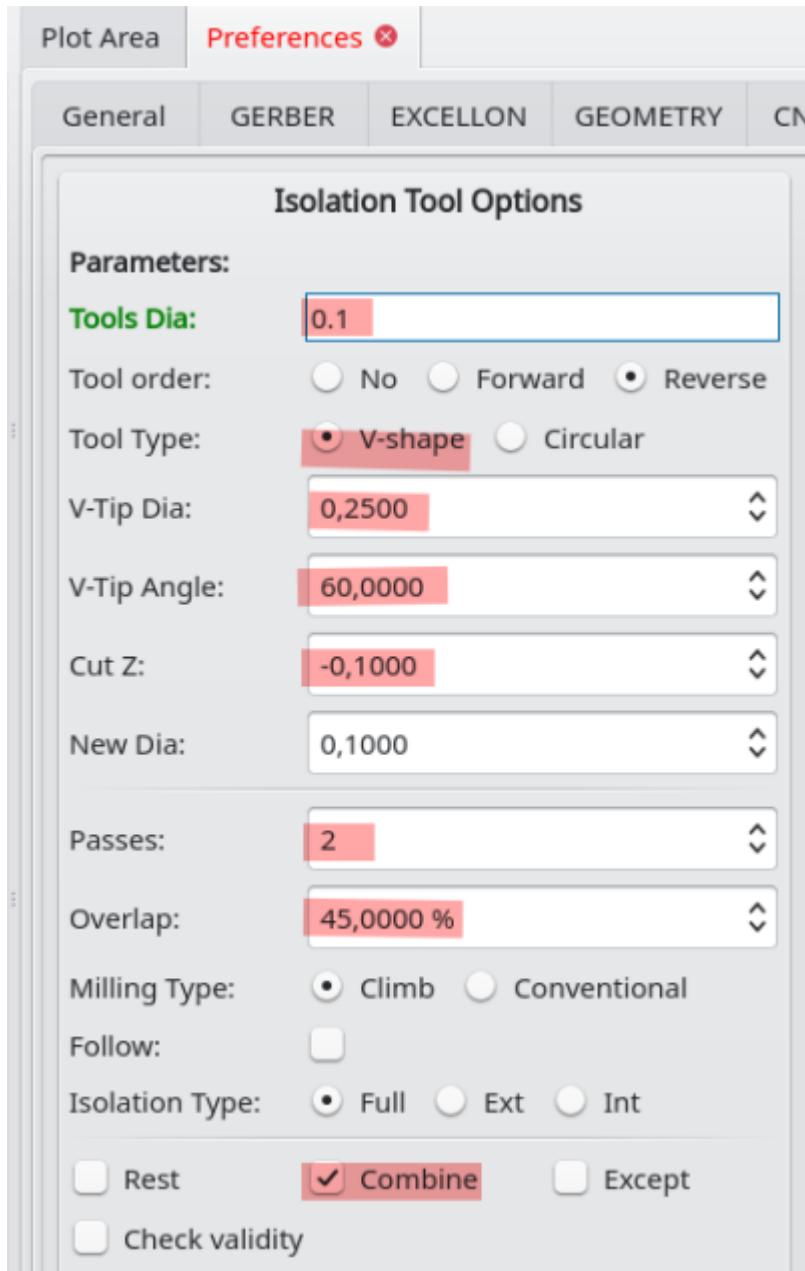


export gerber files



flatcam

setup flatcam



Cutout Tool Options

Parameters:

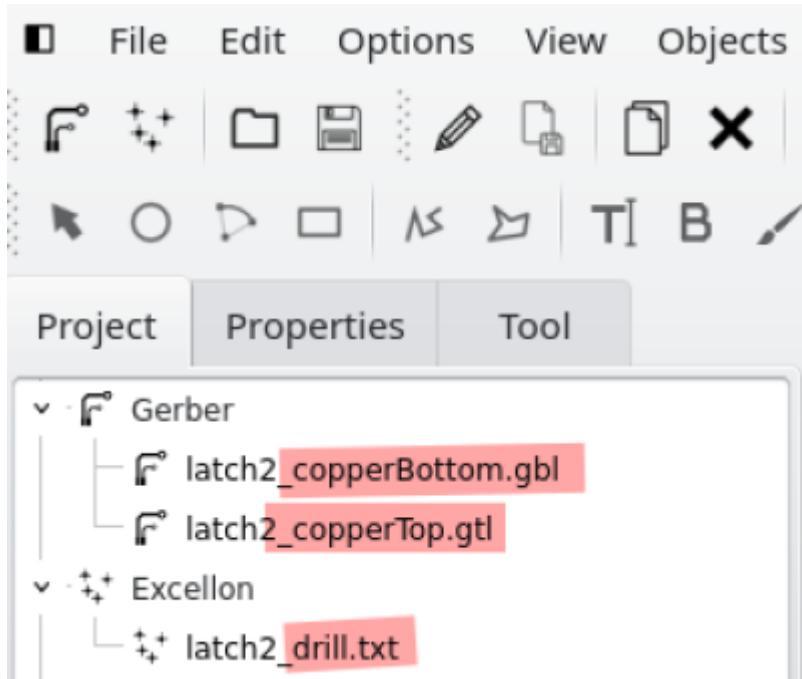
Tool Diameter:	2,0000
Cut Z:	-2,0000
<input checked="" type="checkbox"/> Multi-Depth:	0,2000
Kind:	<input checked="" type="radio"/> Single <input type="radio"/> Panel
Margin:	0,1000
Gap size:	4,0000

Plot Area Tools Database

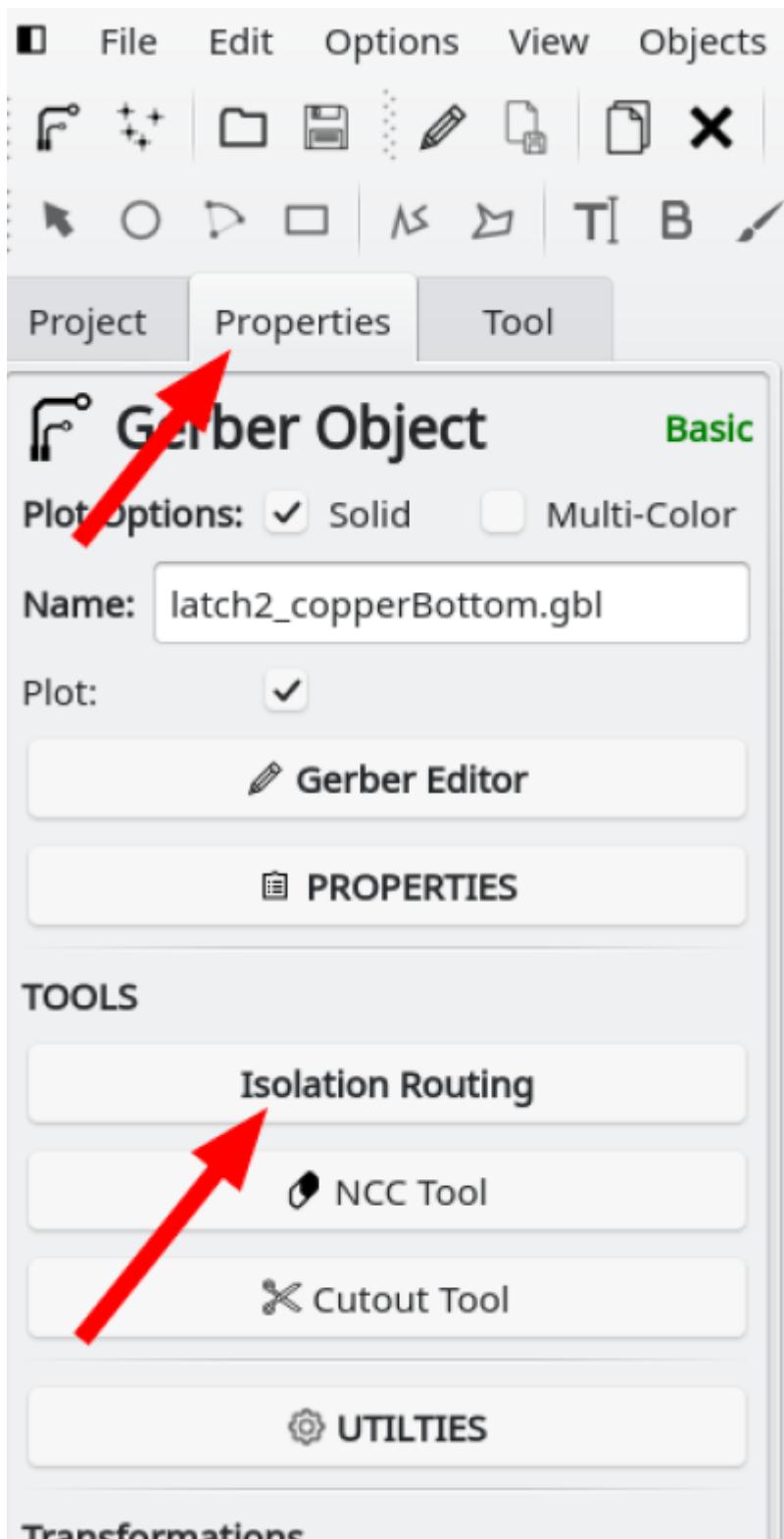
ID	Tool Name	Tool Description	Drilling Parameters	Milling Parameters
1	v-tool 60 0.1	Name: v-tool 60 0.1 Diameter: 0.1924 Diameter Tolerance: Min: 0.0000 Max: 0.0000 Operation: General	Cut Z: -2,0000 Offset Z: 0,0000 MultiDepth: <input checked="" type="checkbox"/> DPP: 0,2000 Travel Z: 2,0000 Feedrate Z: 50,0000 Feedrate Rapid: 500,0000 Spindle speed: 5000 Dwell: <input type="checkbox"/> Dwelltime: 0,0000 Drill slots: <input type="checkbox"/> Overlap: 0,0000 % Last drill: <input checked="" type="checkbox"/>	Shape: V V-Dia: 0,1000 V-Angle: 60,0000 Tool Type: Rough Tool Offset: Path Custom Offset: 0,0000 Cut Z: -0,0600 MultiDepth: <input type="checkbox"/> DPP: 0,0000 Travel Z: 1,0000 ExtraCut: <input type="checkbox"/> E-Cut Length: 0,1000 Feedrate X-Y: 60,0000 Feedrate Z: 60,0000 FR Rapid: 500,0000 Spindle Speed: 6666,0000 Dwell: <input type="checkbox"/> Dwelltime: 1,0000

calcolato

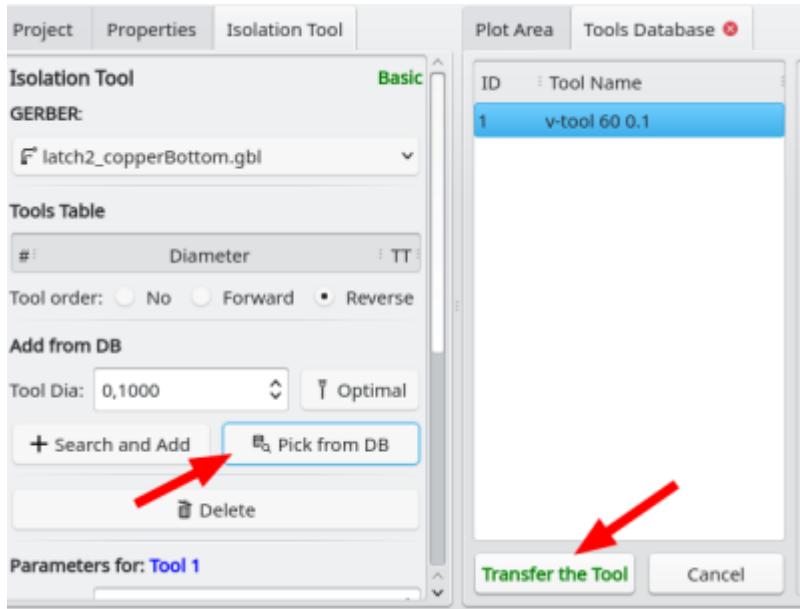
open gerber files (top and bottom if present) and exellon drill file



select copper top and



delete tool from tool table 'pick from db' and 'generate geometry'

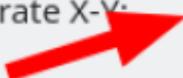


generate cnc job object

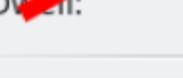
Project Properties Isolation Tool

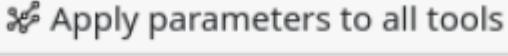
Parameters for: Tool 1

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





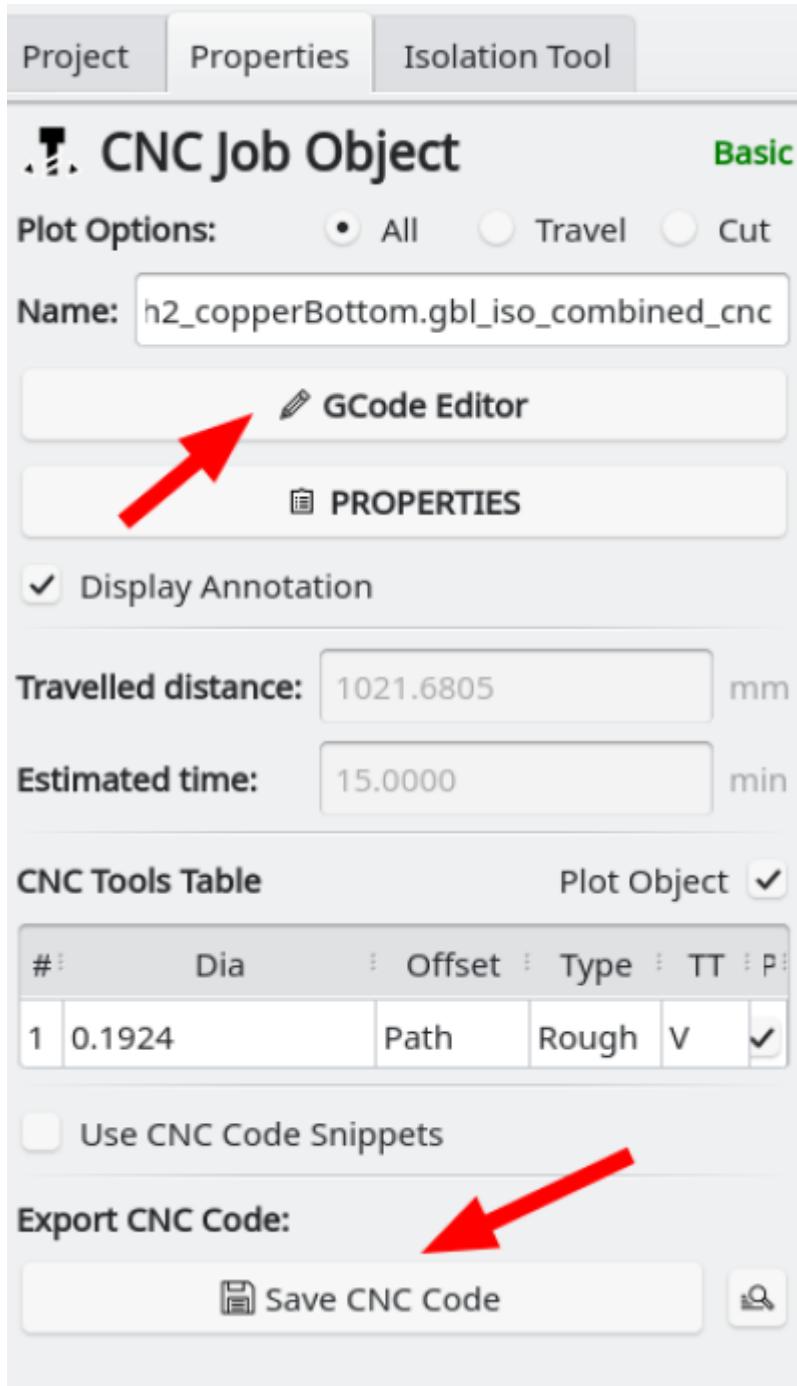




Common Parameters

Tool change Z: 15,0000
End move Z: 15,0000
End move X,Y: None
Preprocessor: default
 Add exclusion areas
 Add Polish


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.

Project Properties 2-Sided Tool

2-Sided PCB

Source Object:

Type: Gerber Geometry Excellon

Bounds Values:

X min: 0,0000

Y min: 0,0000

X max: 0,0000

Y max: 0,0000

Centroid: Center point coordinates

Mirror Operation:

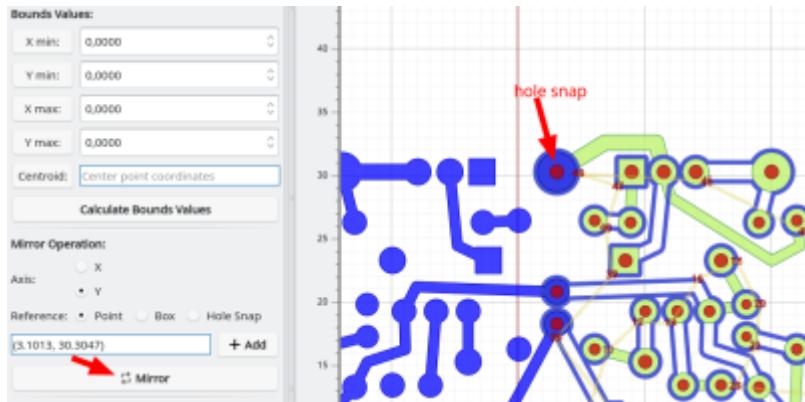
Axis: X Y 1

Reference: Point Box Hole Snap 2

(3.1013, 30.3047)

Excellon:

3



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

 UTILITIES



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

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

carbide3d

set spindle 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*_etch_copper_bottom_mirror.svg
 - yourfilename*_etch_mask_bottom_mirror.svg

From:
<https://wiki.csgalileo.org/> - **Galileo Labs**



Permanent link:
<https://wiki.csgalileo.org/tips/pcb?rev=1620627045>

Last update: **2021/05/10 08:10**