

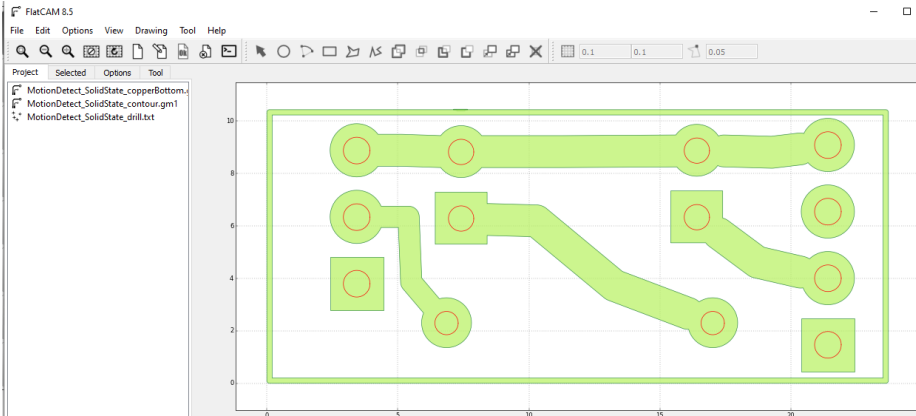
PCB isolation routing using FlatCAM and ESTLCAM

Prerequisites

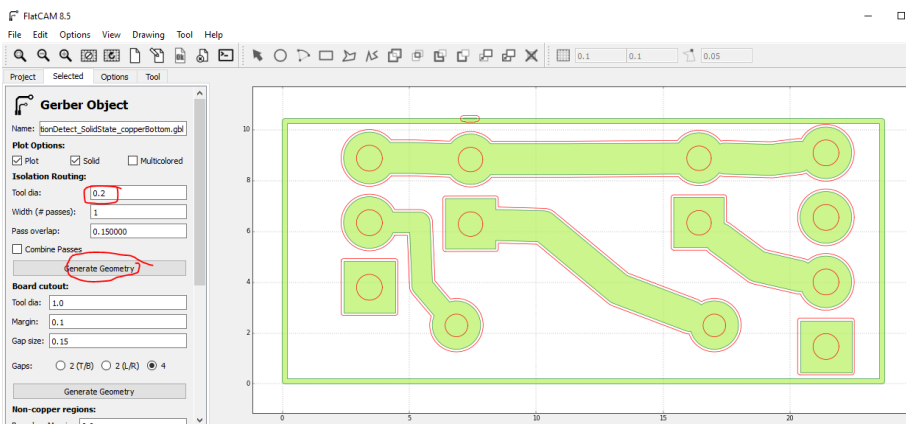
1. You know the general milling process (CAD Design -> Toolpath generation -> GCODE -> Setting up CNC -> Milling)
2. You already have your GERBER Files for the PCB, if not check out Fritzing software to design your electronic circuits. (<https://fritzing.org/>)
3. You have FlatCAM installed (>= Version 8.5)
<https://bitbucket.org/jpcgt/flatcam/downloads/>
4. You know how to work with ESTLCAM (otherwise check the videos on their Web-Page:
<https://www.estlcam.de/anleitung.php>)

In FlatCAM

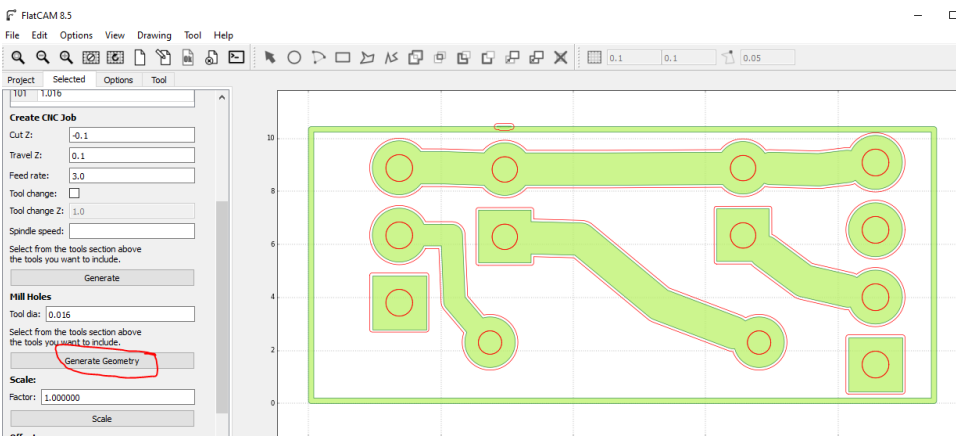
1. Goto File-> Open Gerber then select the outline and the bottom side of the board
2. Goto File-> Open Excellon then select the drilling

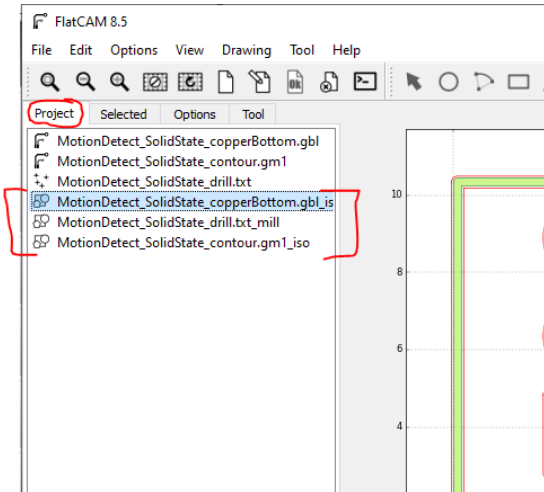


3. Select the bottom-side file in the «Project» tab then change to the «Selected» tab, choose 0.2 as tool diameter and then click «Generate Geometry»
4. Do the same for the file with the outline



5. Select the drill-file, go to the «Selected» tab and click «Generate Geometry» in the Mill Holes Section



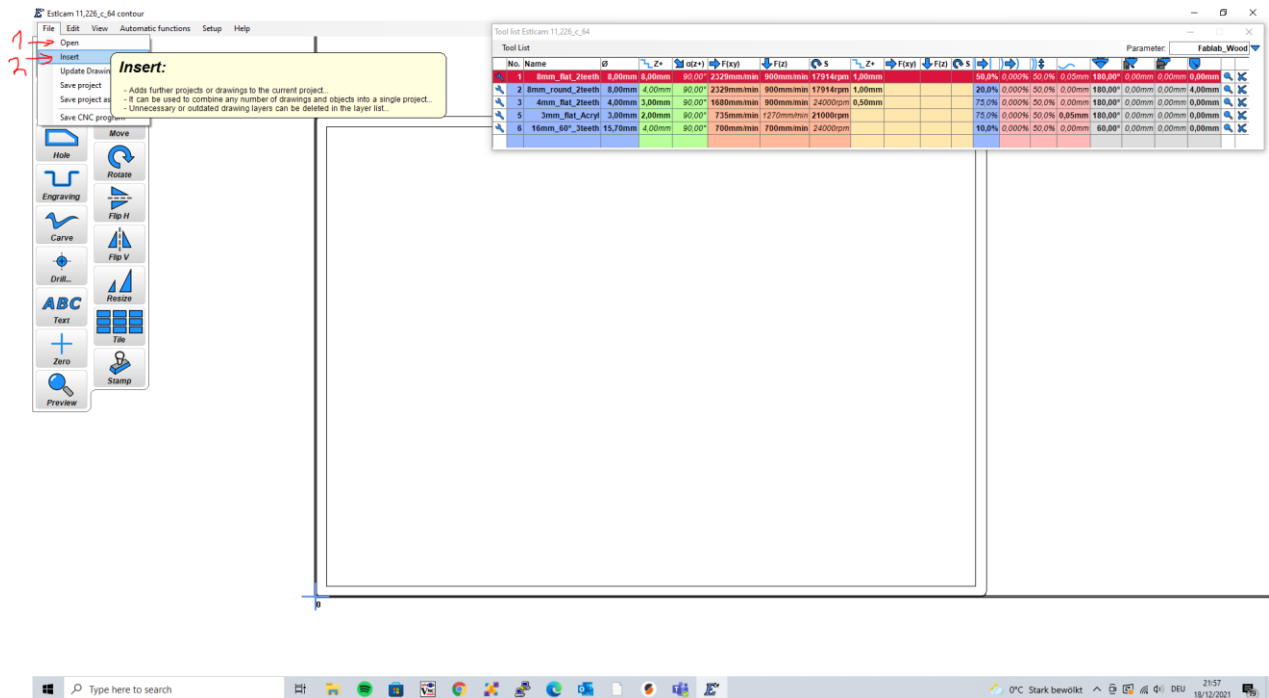


You generated 3 new files (see red brackets left) which contain the geometry for the box, the pcb-routes and the holes.

Goto File -> Export SVG for each individual file and save the new files for further processing in **ESTLNCAM**

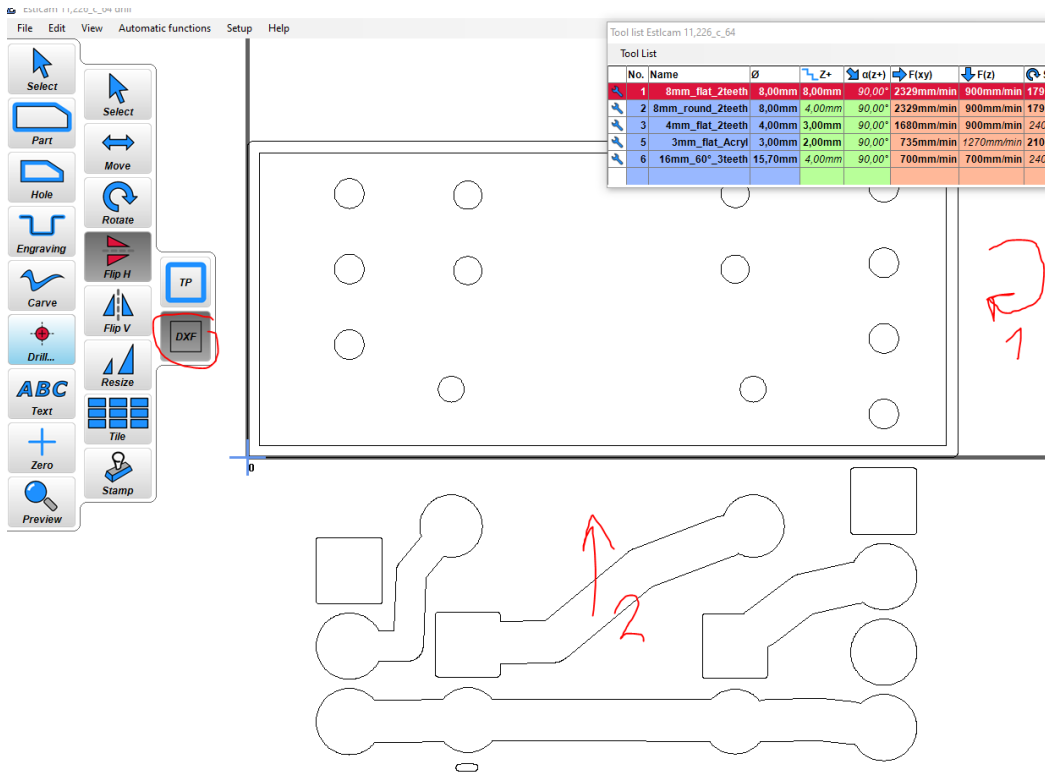
In ESTLNCAM

1. Goto File -> Open and select one of the previous exported files
2. Goto File -> Insert and select the remaining 2 files (one after the other)



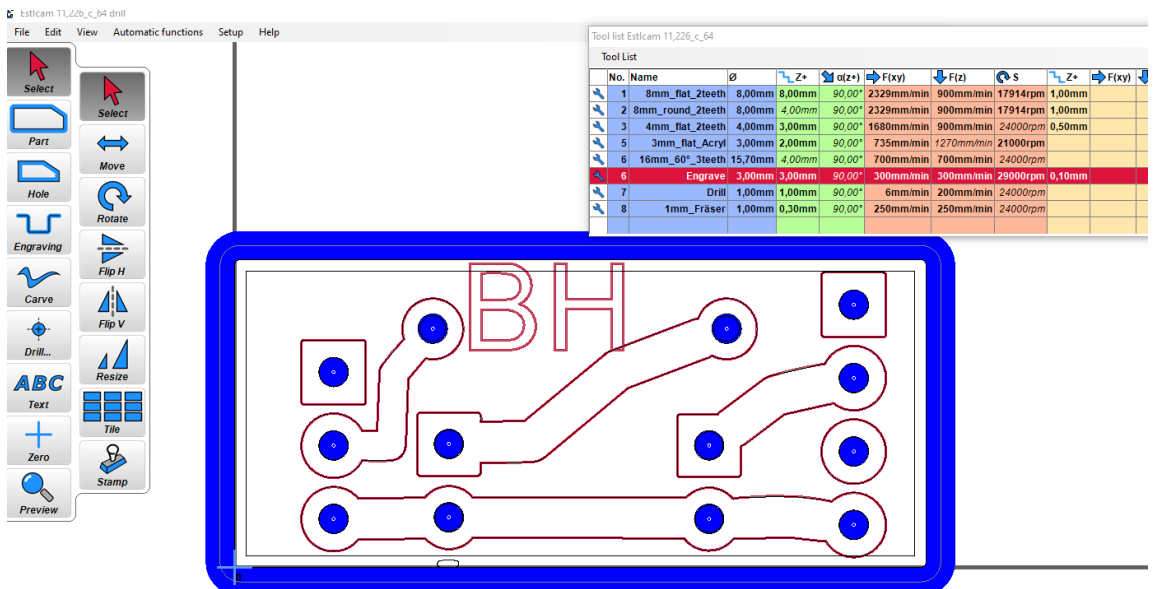
In ESTLCAM

1. The bottom routes will appear mirrored which is correct because they are processed from the bottom side of the PCB. Since you don't want to do the cutout and the drilling from the top-side, you have to mirror those geometries (see 1)
2. Afterwards just move the PCB routes on top of the mirrored holes



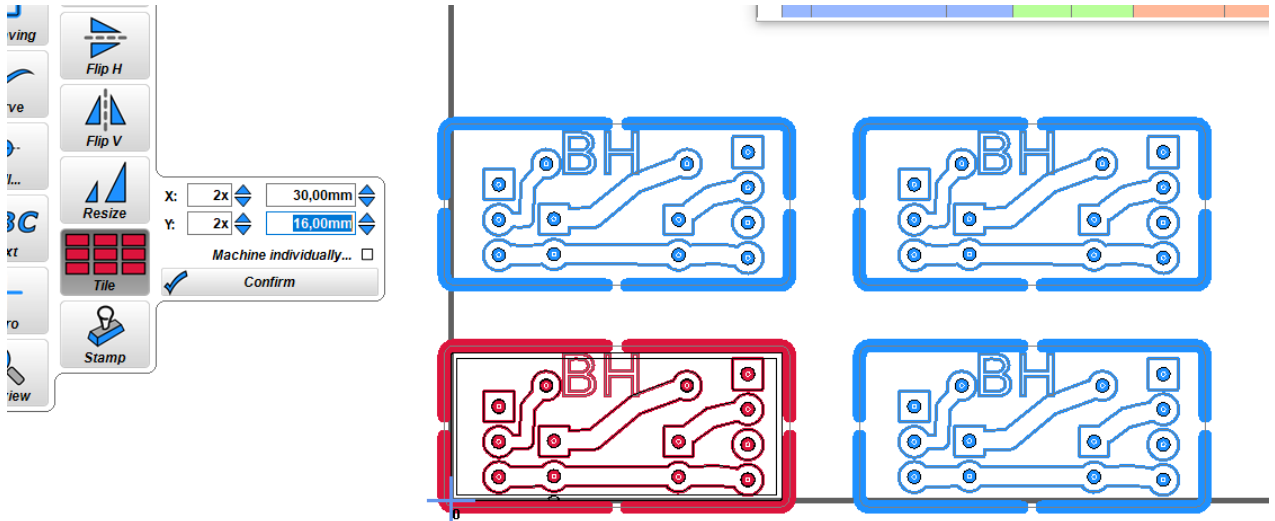
3. When you are happy with your routing path, go to File -> SAVE CNC program

The big advantage of generating the GCODE in ESTLCAM is, that you can work with your tool library as you're used to and you can do a simulation of the routing itself. (Click «Preview»)



In ESTLCAM

Also modifying the drawing are creating multiple copies is very easy in ESTLCAM



On the PCB mill

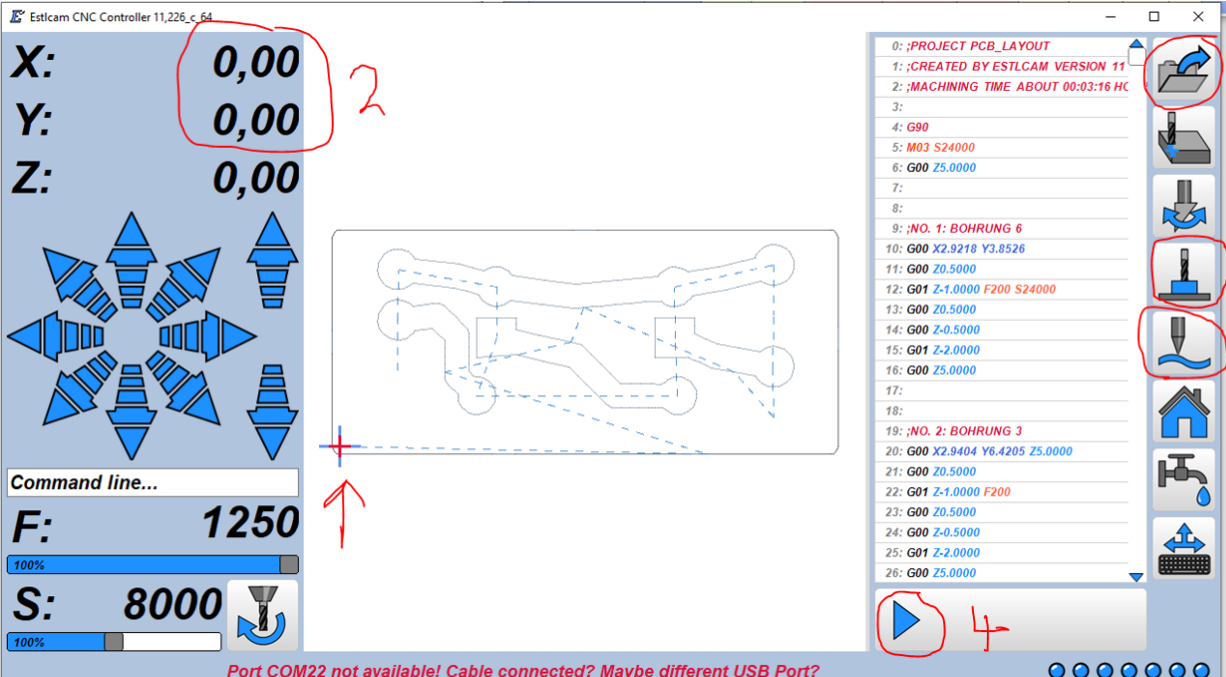
1. Fit the first tool in the Proxxon drill
2. Place the PCB on the magnetic holders, use the screw type holders on the opposite corners
3. Clamp the red crocodile clamp on the first tool and the aluminum block on the PCB (When the tool touches the PCB the circuit is closed for the tool-length measurement but be careful, **NOT ALL TOOLS** are **CONDUCTIVE**, better check before trying a new tool)



In ESTLCAM goto View-> CNC controller to open the machine controller

In ESTLCAM CNC Controller

1. Open the file saved in the previous ESTLCAM operation
2. Move the machine to the starting position of the PCB board and right-click X & Y to set them to zero.
3. Make sure the aluminium block and the red-clamp are fixed properly and start the surface scan, after this operation, the high Z will be set correctly.
4. Afterwards hit the play icon to start milling.
5. After a while you will be requested to change the tool (e.g. from drilling tool to carving tool) **Make sure to adjust the tool length** before hitting play again.



Estlcam CNC Controller 11,226, c. 64

X: 0,00
Y: 0,00
Z: 0,00

Command line...
F: 1250
S: 8000

Port COM22 not available! Cable connected? Maybe different USB Port?

```

0: ;PROJECT PCB_LAYOUT
1: ;CREATED BY ESTLCAM VERSION 11
2: ;MACHINING TIME ABOUT 00:03:16 HC
3:
4: G90
5: M03 S24000
6: G00 Z5.0000
7:
8:
9: ;NO. 1: BOHRUNG 6
10: G00 X2.9218 Y3.8526
11: G00 Z0.5000
12: G01 Z-1.0000 F200 S24000
13: G00 Z0.5000
14: G00 Z-0.5000
15: G01 Z-2.0000
16: G00 Z5.0000
17:
18:
19: ;NO. 2: BOHRUNG 3
20: G00 X2.9404 Y6.4205 Z5.0000
21: G00 Z0.5000
22: G01 Z-1.0000 F200
23: G00 Z0.5000
24: G00 Z-0.5000
25: G01 Z-2.0000
26: G00 Z5.0000
  
```