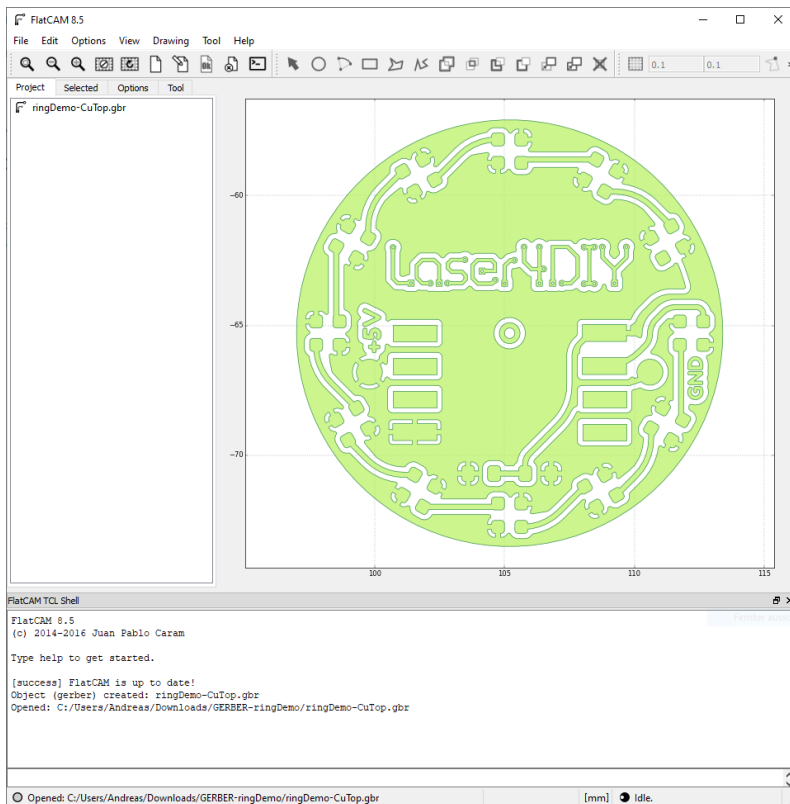# Using Laser4DIY for Prototyping PCBs

Creating Tool Paths

The first step is to create tool paths to be run on the Laser4DIY device. This is very similar to isolation milling, we just replace the milling tool with a laser. Therefore, we can use software suitable for PCB isolation milling. In this tutorial, we use FlatCAM, an open source tool, which can also import Gerber files. There is also a tutorial on the FlatCAM site.

Import GCode

- Open FlatCAM
- File Open Gerber
- Load the top CU layer

Setup Tool

- Select Gerber layer in project tab
- Switch to 'Selected' tab
- Set tool dia to 0.05
- Set to 1 pass
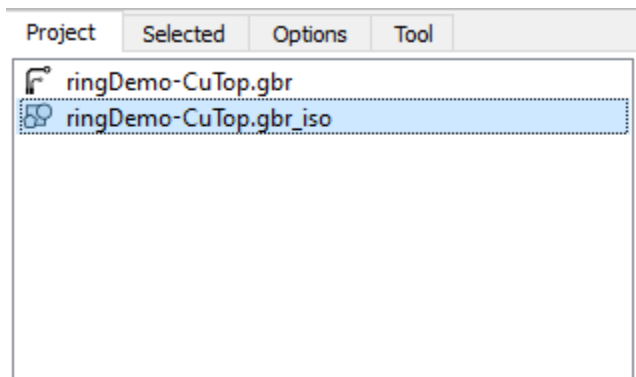
Create Geometry

- Press the "Create geometry" button
- The red lines around every trace indicate where the laser will ablate copper
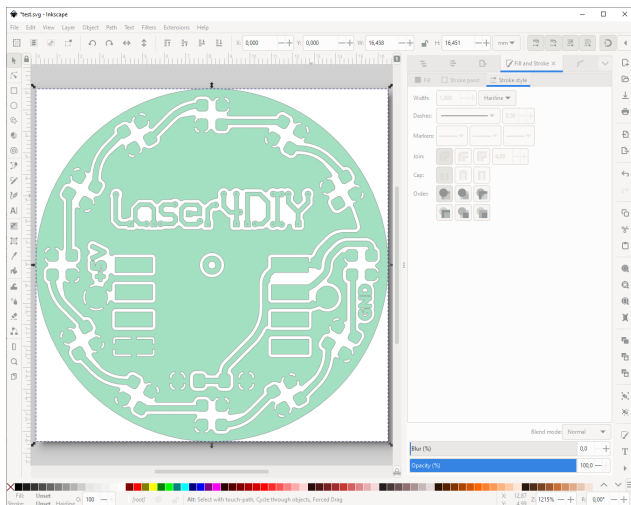

Save SVG and Clean it Up

- Go back to the Project tab
- there now should be a new "_iso" entry

- Select it
- Choose File Export SVG
- There now should be an SVG file exported with the tool path we just generated
- Open the SVG in Inkscape to check if everything worked as expected
- When you only see a blank white canvas, it is possible that all geometry is outside the document. To fix that
  - press 4 (or select View Zoom Zoom Drawing) to change the viewport to the drawing
  - press Ctrl+A to select everything
  - press Ctrl+X to cut it
  - press 5 to view the document
  - Ctrl+V to paste it
  - Move the drawing on the document, make sure snapping mode is on
- Make all lines hairlines
  - press Ctrl+A to select everything
  - Select Object Fill and Stroke...
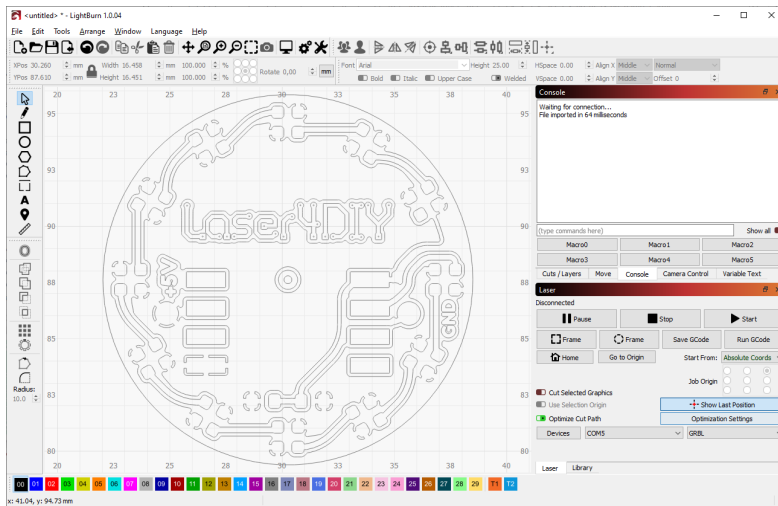 - In Stroke Style tab select Hairline
- Save SVG again

Running ToolPath in Lightburn

The Laser4DIY device can be controlled by any CNC software that supports laser cutters running Grbl. We are using Lightburn. The following steps are Lightburn specific, for other software packages, corresponding steps have to be performed.

Lightburn initially has to be configured to use the Laser4DIY device. Please refer to this guide for doing so.

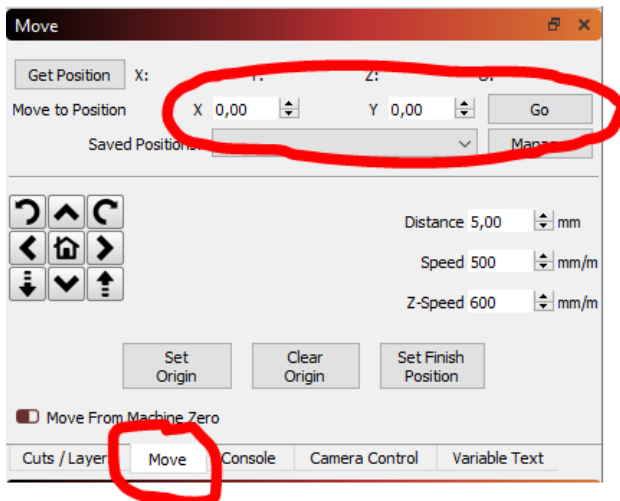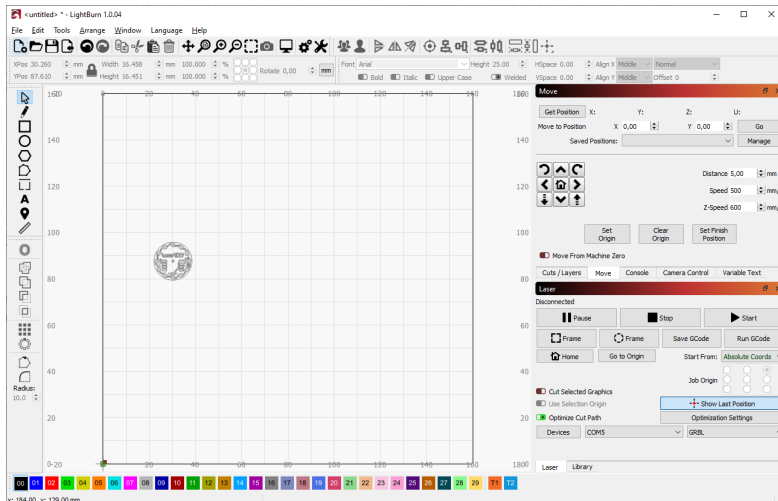### Loading SVG

The generated toolpath can simply be loaded by drag-n-dropping it into the workspace:
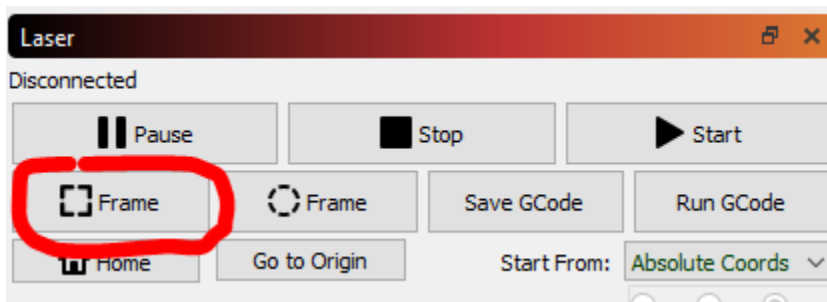
Placing Toolpath at Right Position

- You can move the toolpath to the correct position (where the PCB material lies on the XY table) by simply dragging it
- In the 'Move' section you can enter coordinates to check if the positioning is correct. Use coordinates shown on the workspace view, press the Go button and check if the table to the position you expected.

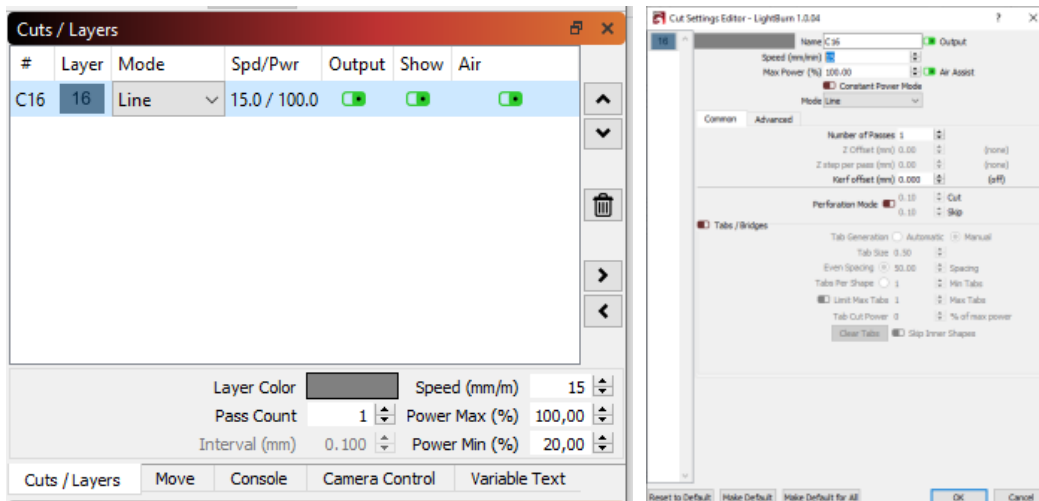- Alternatively you can use the "Frame" feature

- Please note that the Laser4DIY device does not have a pilot laser. Positioning by eye cannot be done exactly therefore.
- We recommend using a PCB blank a bit larger than needed to compensate for small placement offsets.
- If you need exact placement (e.g. for two sided PCBs), you can use the stops of the table. Please make sure you have adjusted the end stops precisely if you want to rely on them

Setting Speed

The output power of the Laser4DIY device varies, depending on how much power your seed laser diode emits (they are all a bit different) and how well your device is tuned. Because of that the ablation speed needs to be adjusted for your device. A speed of 15 mm/min worked well for us, but we recommend doing some small tests before starting a big PCB job. Doing 2 faster passes (2 passes with 30 mm/min each) can produce better results, but in this case backlash compensation has to be tuned very precisely so that the second pass ablates at exactly the same position again.
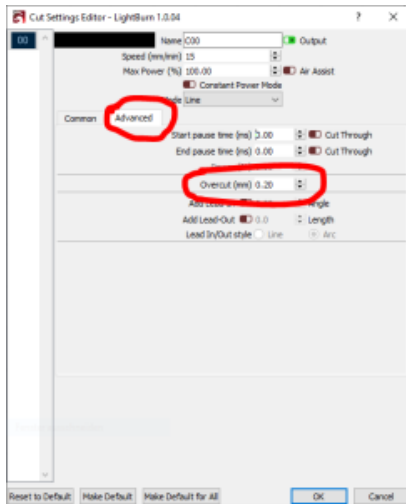
In Lightburn, you set the speed in the Cut/Lasers tab:

By double clicking the speed, another dialog comes up, allowing to change the details. Leave the power setting at 100%, smaller values will not actually change the laser power. Also leave on "Output" and "Show". "Air" does not have any effect.
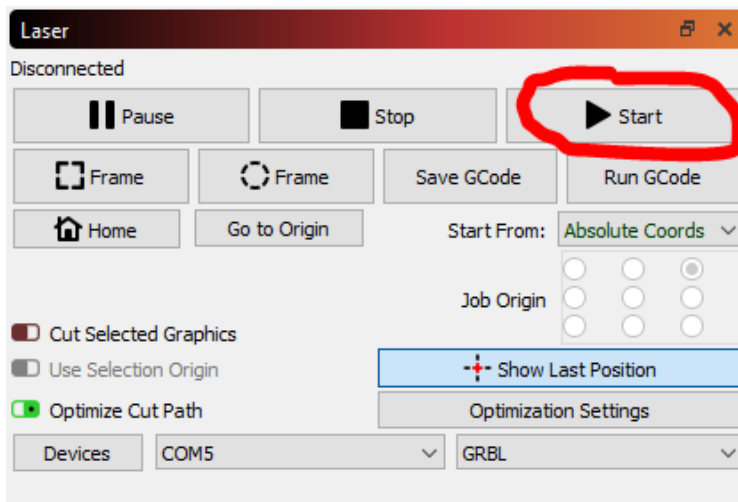
Cutting Closed Paths

To ensure that there are no gaps in the ablation paths, we recommend using the "Overcut" feature. When engraving closed shapes, as it is typically the case for all isolation paths on a PCB, this will continue engraving past the end of the (starting another round of the closed path) for a small amount. Without this setting there could be a a small gap between the start and the end of the loop, causing an electrical connection, which is unwanted of course. The amount of the overcut only needs to be a fraction of a mm, we use 0.2mm.

Running the job

Finally, close the lid of the Laser4DIY and start the job. This is done by hitting the Start button

Lightburn will display a progress bar for the ablation process