

Instructions for making our TinyFABISB using ROLAND MODELA (MDX-50) milling machine

FABLAB BRIGHTON 2018



1. Circuit design - creating RML files	2
2. Setting up MDX-50 Machine	5
Fix board	5
Setup machine	6
To select the tool	6
Zero X and Y	6
Set Z origin	7
Testing depth of cut	9
Safer to leave spindle running on Z up in case of user error	9
Instructions for changing the tool in the machine	10
3. (FabModules) Prep file for traces milling	10
4. Set up machine to cut out board outline	17
5. (FabModules) Prep file for cutting outline	18
Success!	18
6. Cleaning up the board	19

1. Circuit design - creating RML files

You can do all this on your own computer - you don't have to use the machine computer.

You will need to create two separate RML files, one for the TRACES, one for the OUTLINE. This is because they will require different cutters and slightly different settings. Traces is what we will mill out for our circuit, then outline is what will happen last to cut the board out from the board material.

Create/import your TRACES or OUTLINE in photoshop/gimp/etc. One good way to do this is to have a single illustrator file with two layers, one for TRACES, one for OUTLINE. Then you can simply save out each layer in turn as a separate .png file (with a high resolution of 1000 dpi).

Before saving, check that your images are the correct size for your circuit board (in mm, not dpi).

Note: whatever is white will stay, black will be machined. You may need to invert out image to get the white lines and black background.

I.e. White: material left behind will be the conductive copper (tracks)

Black: will be the space around traces, some of this will be removed (not all).

(So for example, your board outline design should look like a black line around a white box)

Note: you should include your own logo in the gap!

You can orient the circuit board based on the orientation you want to cut it on the mill, landscape or portrait.

Illustrator note, export as, on quality, select other and manual 1000 dpi

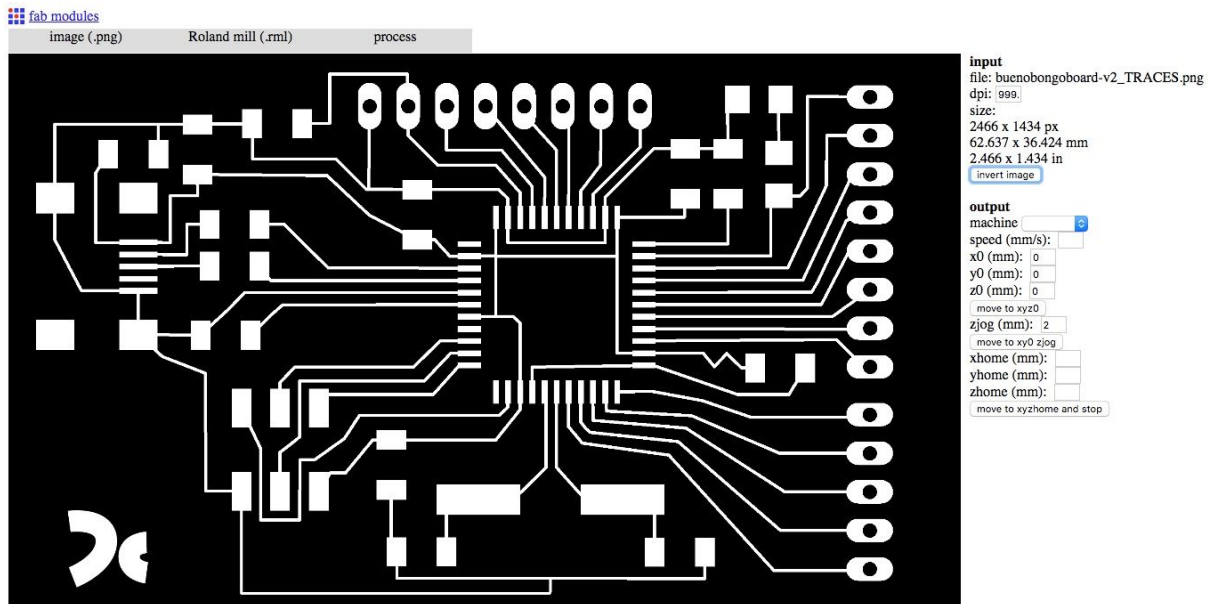
Save as YOURNAME traces and outline.png

Then ALSO save as YOURNAMEtraces.png

Then ALSO save as YOURNAMEoutline.png

Then goto fabmodules.org and use the settings as outlined below. Fabmodules can be used to control a range of machines independently from the machine's own software.

Input format: PNG (do the YOURNAMEtraces.png first, then the YOURNAMEoutline.png). Make sure you INVERT your traces image, so that the space around your traces is BLACK, and the TRACES are WHITE. Here is what the traces should look like, this is the board designed by our instructor Luiz.



Output format: select Roland mill (.rml) (note MDX40 is similar to our machine and we can use this)

Process: PCB traces (1/64) which is about 0.4 mm (so 1/32 is about 0.8mm)

Input the settings as outlined below.

Note that the zjog is the safe travelling distances above the board.

Note also, that if in doubt, it's best to mill the air above the board than to mill too far down. Or you can always use soft foam to try it out.

.....perhaps use image for this....

Speed 4mm/s

x=y=z=0

xhome=yhome=zhome=0, or you can make zhome=2 if you want it to stop just above the piece.

zjog=2 (zjog is the safe travelling distance above the board)

Cut depth=0.1mm

Tool diameter=0.4mm (1/64")

Num of offsets=2 (-1=remove all..up to 4...this is like 'padding' like on html)...1=only 1 milled layer around your trace 4=4 milled layers around your traces.

Offset overlap=50% (is overlap between the offsets from the

Path error=1.1pixels

Image threshold=0.5

.....

Calculate the tool path. Check this by zooming, rotating, panning (instructions for doing these things are at the top of the screen). Save this configuration as YOURNAMEtrace.rml.

Then do the same for the outline using your YOURNAMEoutline.png, but for the option PROCESS, use PCB outline (1/32) which is about 0.8 mm.

Terms:

Stock thickness - thickness of the material e.g. 1.7mm

Cut depth - the depth the tool will cut in one go. E.g. 0.6 will take 3 passes to cut 1.7mm

2. Setting up MDX-50 Machine

Fix board

Wash hands, Wash Board.



Put the double sided tape on the back of your board, making sure that it doesn't overlap and try not to touch the sticky bit.



Then stick your board onto the wax spacer (without the wax spacer the mill won't reach the board as it's too high). Overlap the board very slightly with the edge of the wax block to allow us to remove it afterwards.

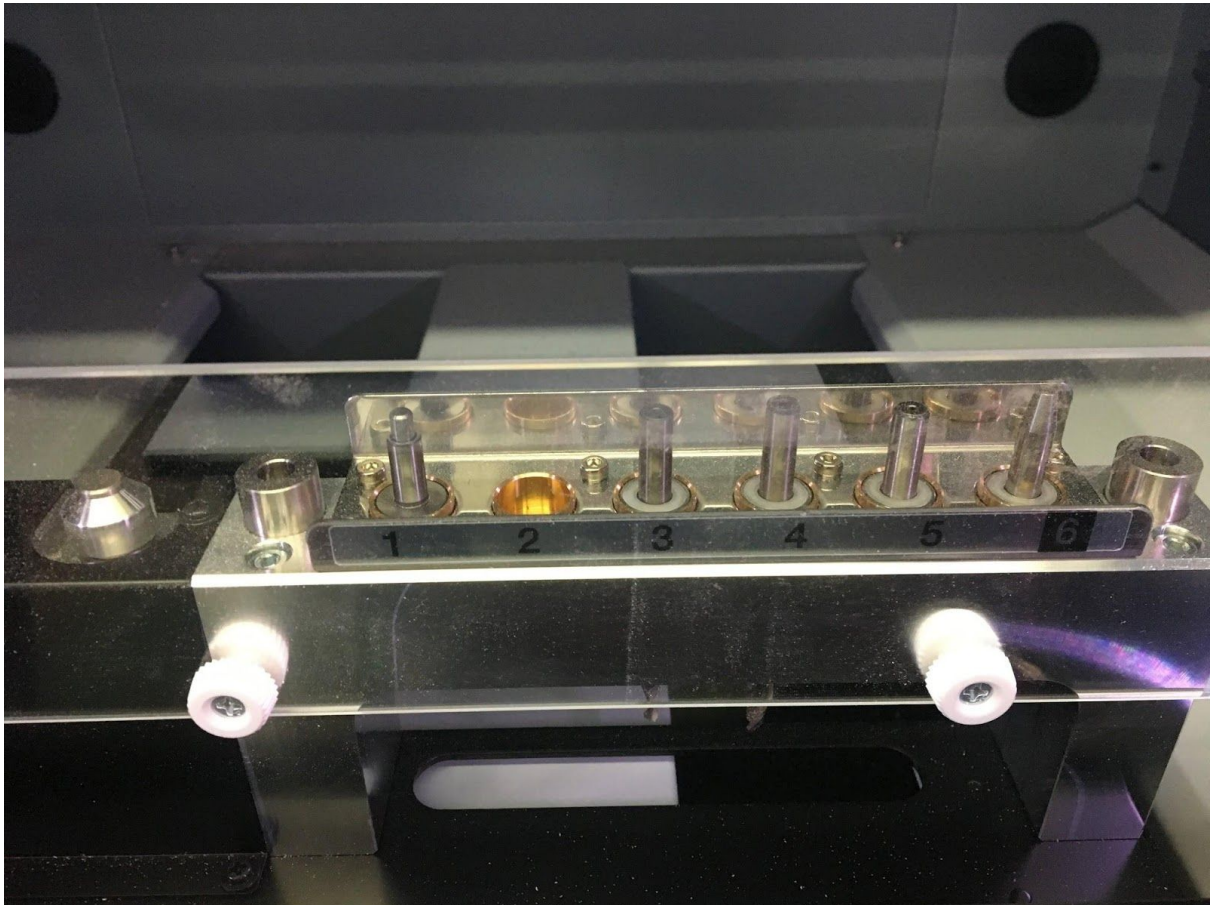
Clean Board again incase of residue.



Setup machine

To select the tool

Know what tool is in what cartridge (1-6).



Press MENU until you reach the TOOL menu, then use the jogwheel to select tool the tool you want (1 or 2 typically). Press ENTER when selected.

At last check, TOOL 1 was for outline, with D=1mm, and TOOL 2, D=0.4mm tool for cutting traces) in this case. If in doubt, check by measuring, or asking.

Tool 2 first (trace), Then tool 1 (outline)

NOTE: we should keep a little table on the desktop/link/label on machine showing what tool is in what slot.

Zero X and Y

Origin is front left of the cutting bed.

Move X and Y with spin knob.

Set origin with: origin + axis

Allow some space around the circuit board - don't set 0,0 to be right in the corner of the copper board.

Press MENU until you reach the user menu, then press X/Y/Z to get the tool closer to the board (for easy sighting), then use the jogwheel to move the tool closer. You can press RATE to move more slowly/quickly to get the tool quite close to the board.

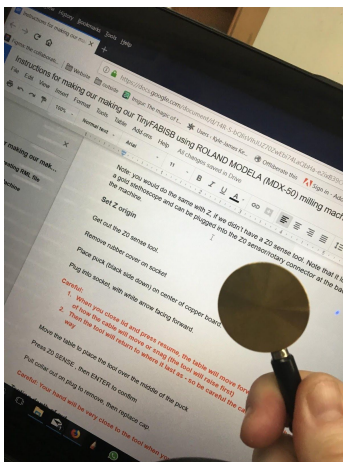
Then press X, use the jogwheel to move it to where you want your X0 to be, then press and hold ORIGIN to set this as your X0 origin value.

Then do the same for Y.

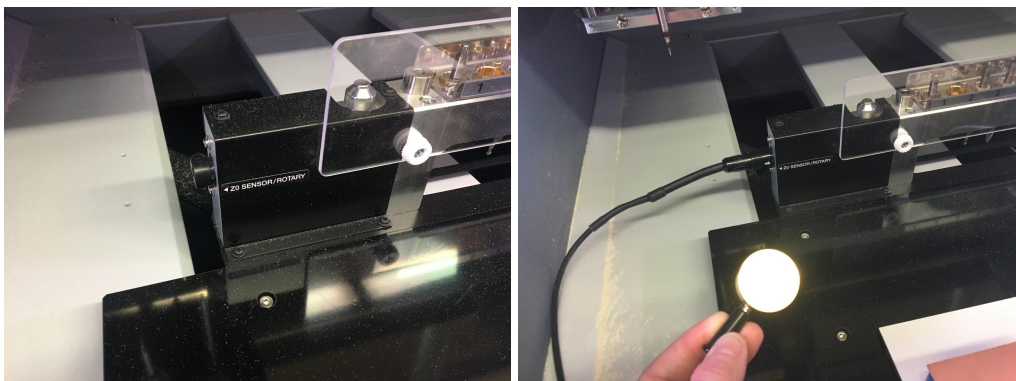
Note: you would do the same with Z, if we didn't have a Z0 sense tool. Note that it looks like a gold stethoscope and can be plugged into the Z0 sensor/rotary connector at the back of the machine. NOTE: when setting Z manually, we have found that setting z0 as approx 0.03mm below the actual surface of the board works well.

Set Z origin

Get out the Z0 sense tool. Clean Z0 Tool



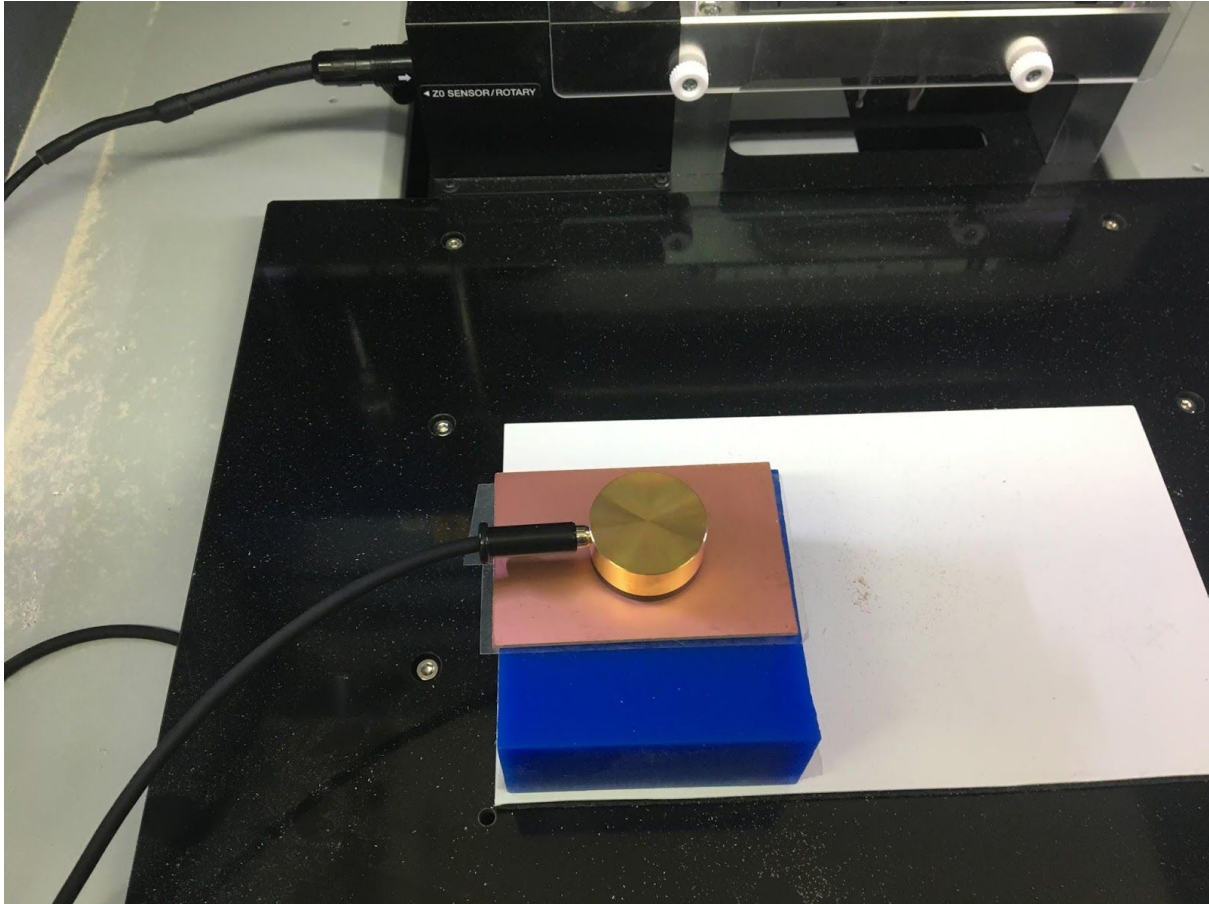
Remove rubber cover on socket. Plug into socket, with white arrow facing forward.



Place puck (black side down) on center of copper board.

Careful:

1. When you close lid and press resume, the table will move forward, so be aware of how the cable will move or snag (the tool will raise first)
2. Then the tool will return to where it last as - so be careful the cable is not in the way



Move the table to place the tool over the middle of the puck

Press Z0 SENSE , then ENTER to confirm



Pull collar out on plug to remove, then replace cap.

Careful: Your hand will be very close to the tool when you do this!

Testing depth of cut

Note: to test the depth of the cut, you can turn spindle on/off, by pressing SPINDLE and hold. You can then rotate the jogwheel to increase the spindle speed. Then you can press Z and move this down slowly with a rate of x10 onto the board.



Safer to leave spindle running on Z up in case of user error



If you want to inspect the piece, Press MENU until you reach the move menu, then go to VIEW this can be used to move the table close to the front.

If you want to, return the tool to the user XY position: Press MENU, the USER XY.

Cycle through the MENU button until you get to the MOVE menu, then go to USER XY menu and press ENTER.

NOTE: the spindle speed for cutting PCB traces and outlines should always be between 12,000-13,000 RPM. THIS IS IMPORTANT!

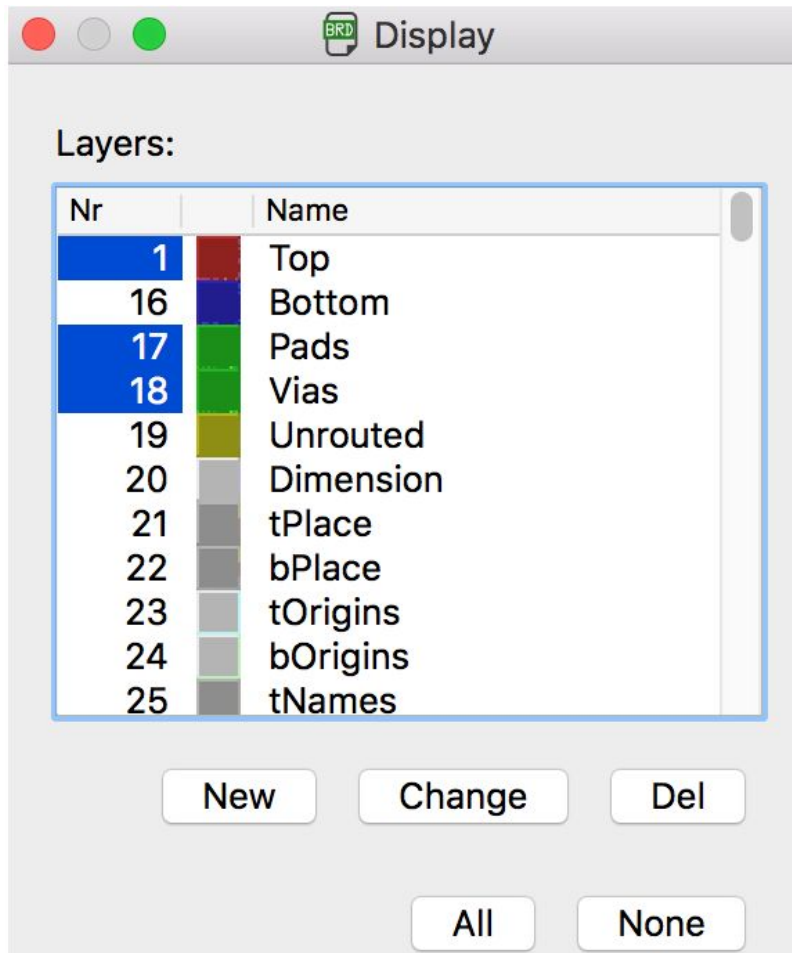
Instructions for changing the tool in the machine

For full instructions on measuring and changing the tool see Des's page:

<http://fab.academany.org/2018/labs/fablabbrighton/students/derek-covill/w10.html>

3. (FabModules) Prep file for traces milling

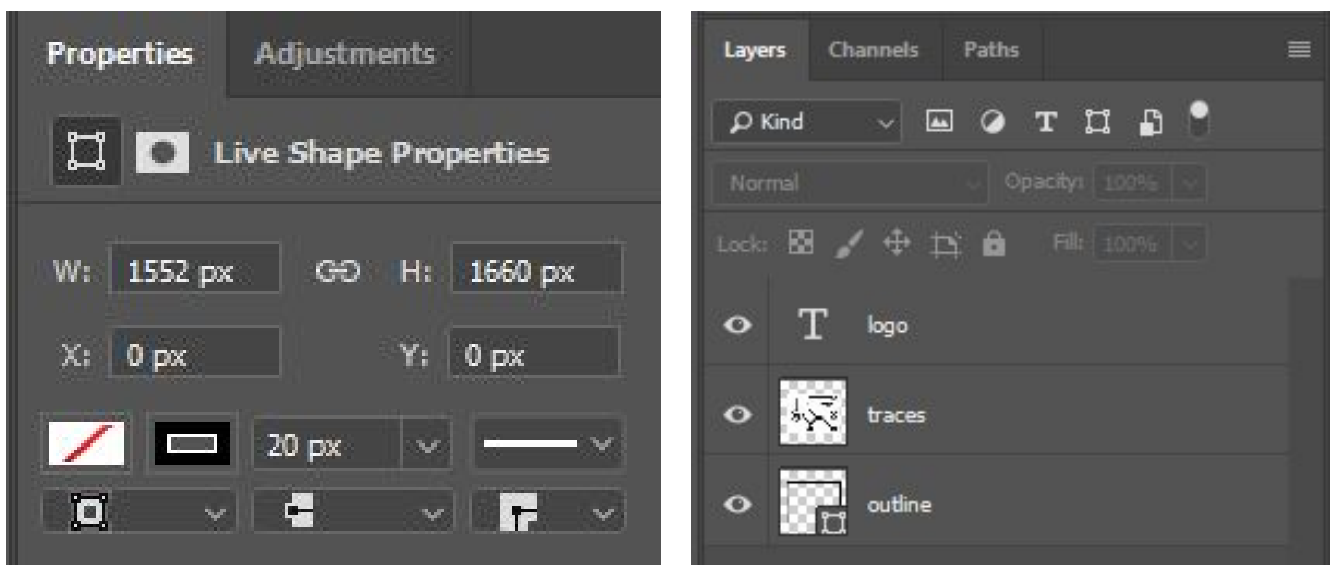
In Eagle, in layers menu in the top toolbar, select only the top layer, pads and vias (traces only) .



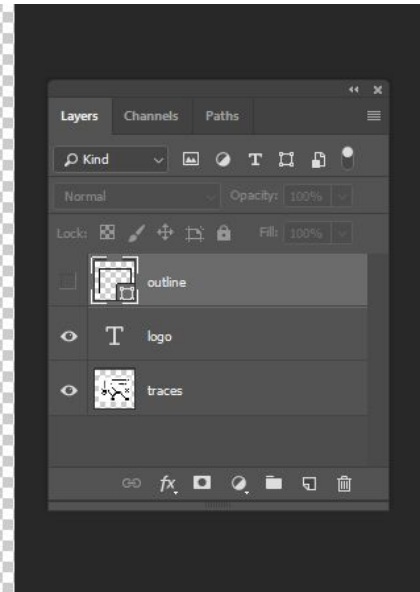
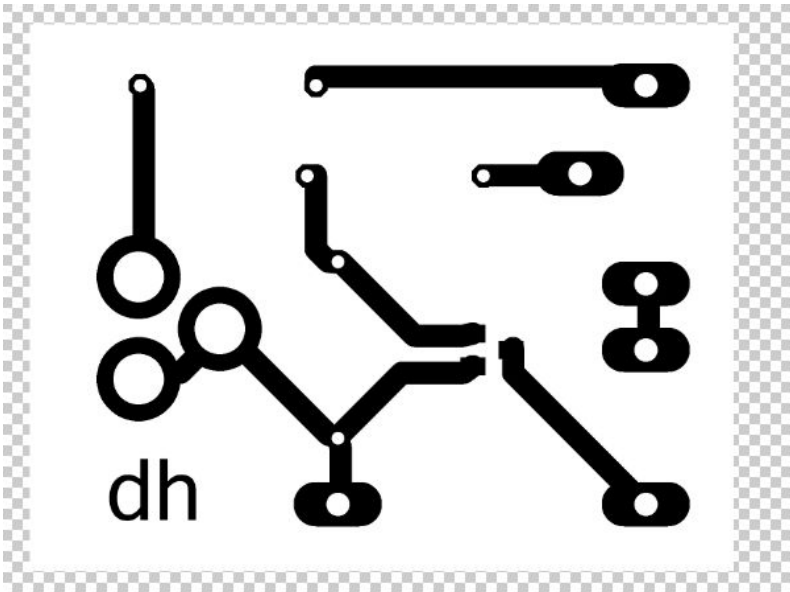
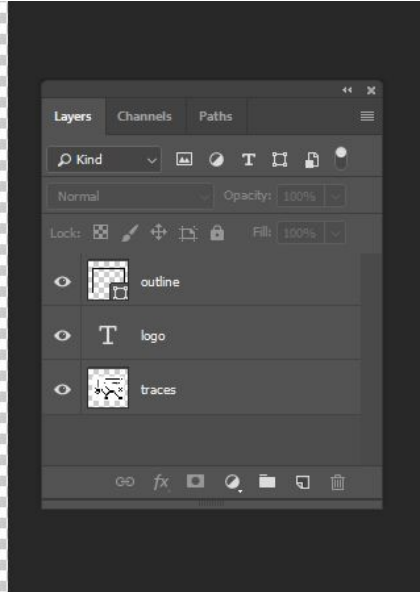
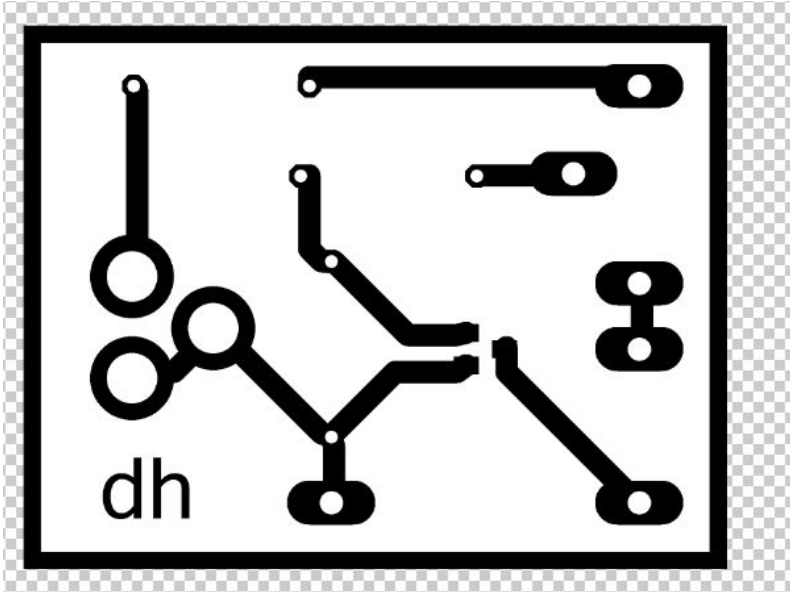
- then export as a png (file -- export -- image)
- settings should be MONOCHROME and 500 DPI - this will export a image with white traces.
- Make sure the color mode is set to greyscale: image > mode > greyscale
- Remember: the Modela cuts out the DARK and leaves white!

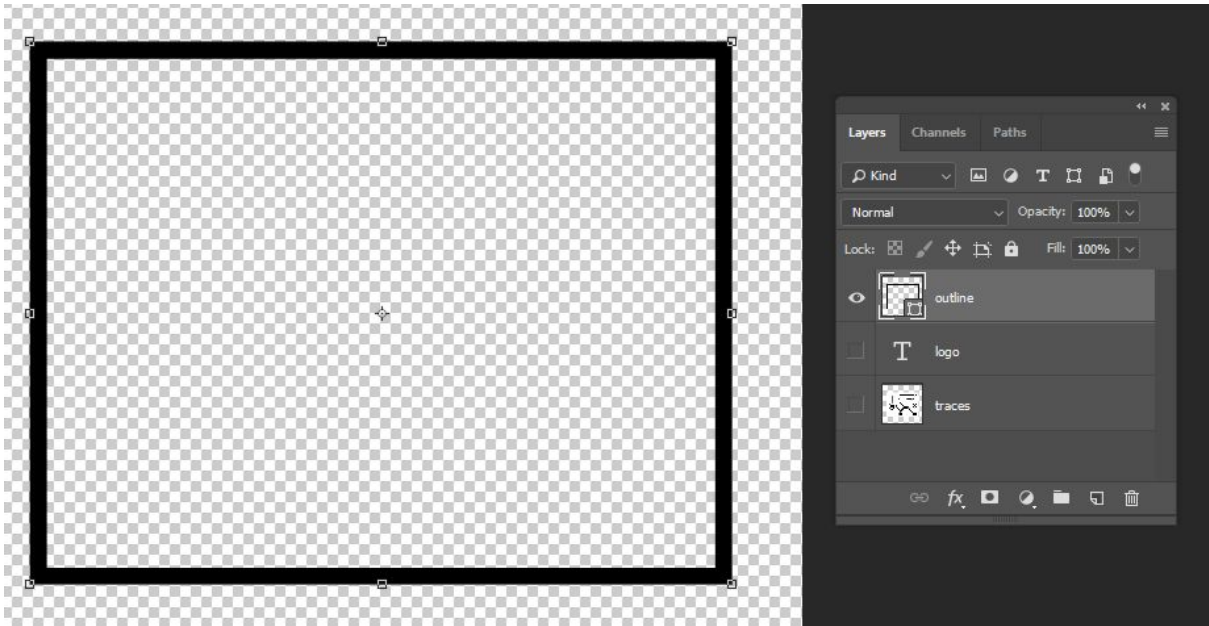
In Adobe Photoshop, open the .png files separately. Then create a new photoshop file and copy and paste the .png files into separate layers. If the boundary is hard to distinguish from the edge of the canvas, you can change the canvas size to be bigger than the images (Image > Canvas Size).

Note the size of your traces .png file (in pixels, e.g mine today was 1552 x 1660 pixels). Use the rectangle tool (shortcut = U) and create a rectangle that size. Use the move tool (shortcut = V) to position it over the traces. Note that you also want the fill to be transparent, and the line to be 40 pixels wide as shown below. Note that the fab tutorial suggested 20 pixels wide, but we found that with our wider cutter for the outline (1.0mm diameter), this sometimes wouldn't generate the g-code, so a wider line of 40 pixels works better.

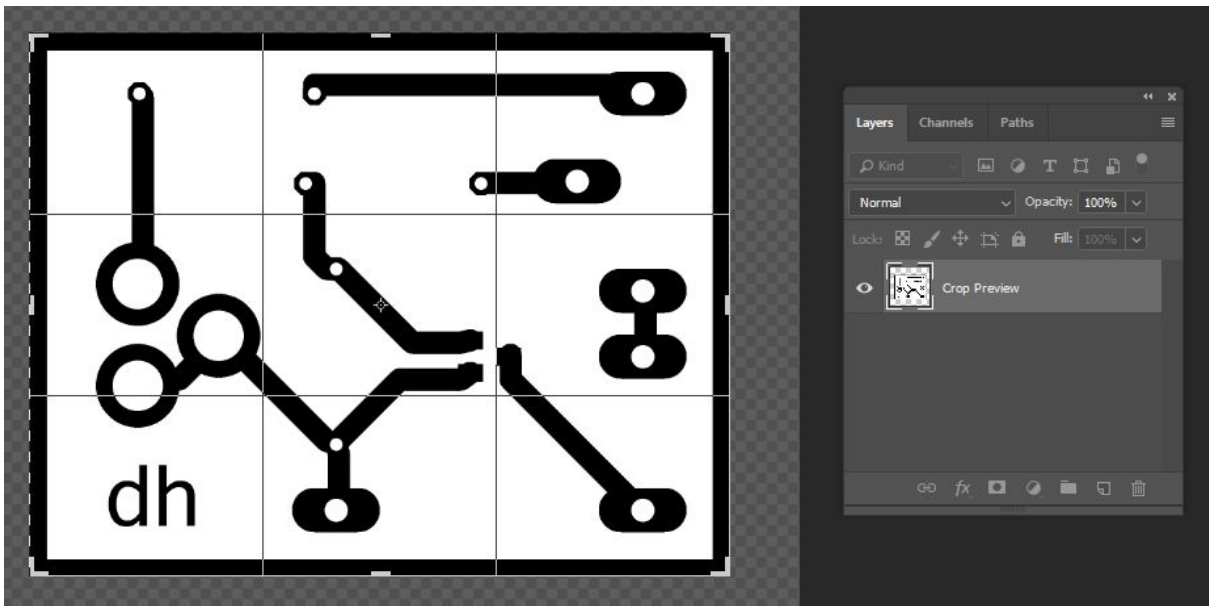


One good reason to use Photoshop or GIMP for this process is that you can then add in a logo to your board. A simple way to do this is to create a new layer with your logo on it, and put this on top of your traces layer. Here we show that process just before saving each png file.

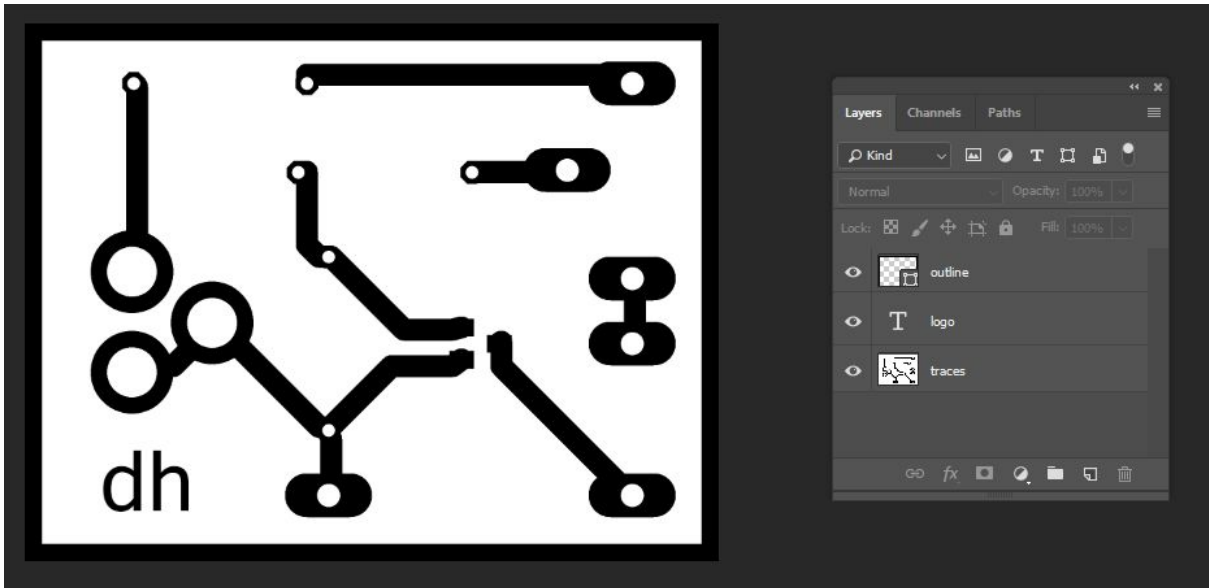




Then it is important to crop the image (crop tool shortcut = C), otherwise the origin on your milling machine will be the bottom left of the CANVAS, NOT the bottom left of the OUTLINE.



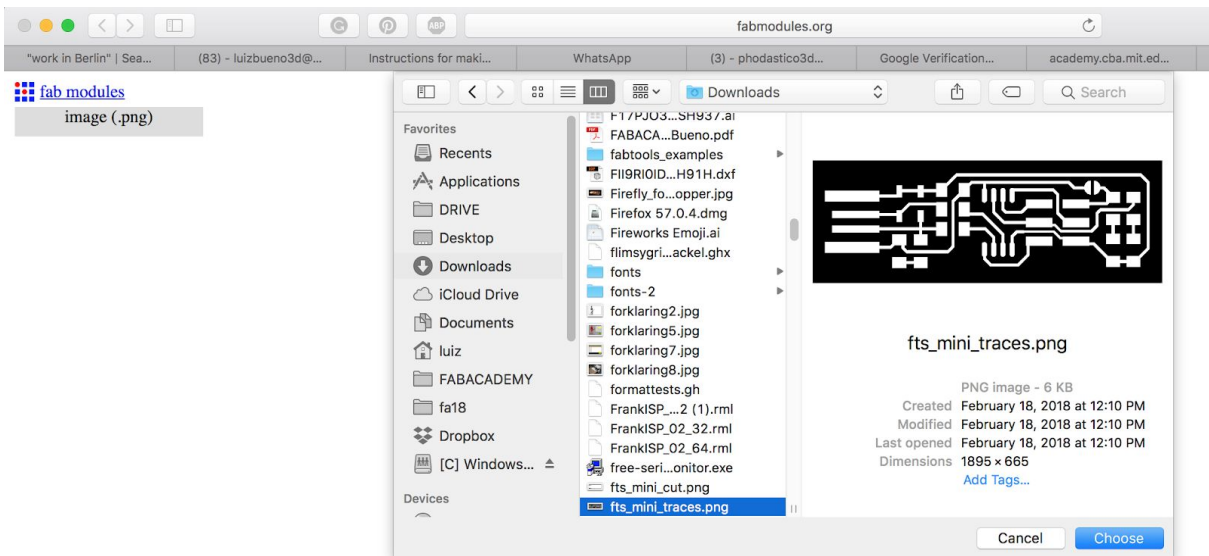
Hit enter when it's cropped.



Then you can show each layer in turn and to save each layer as a separate .png file.

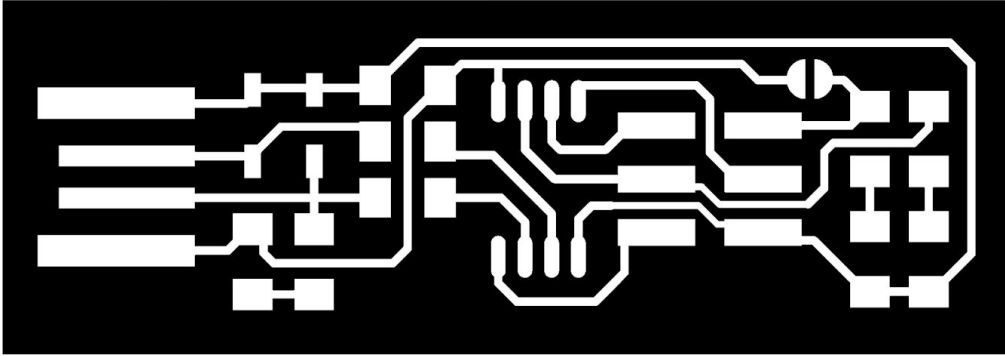
Then save the .psd file for your records...it's a useful file as it contains both both your traces and outlines.

Then goto fabmodules.org and load the traces.png file.



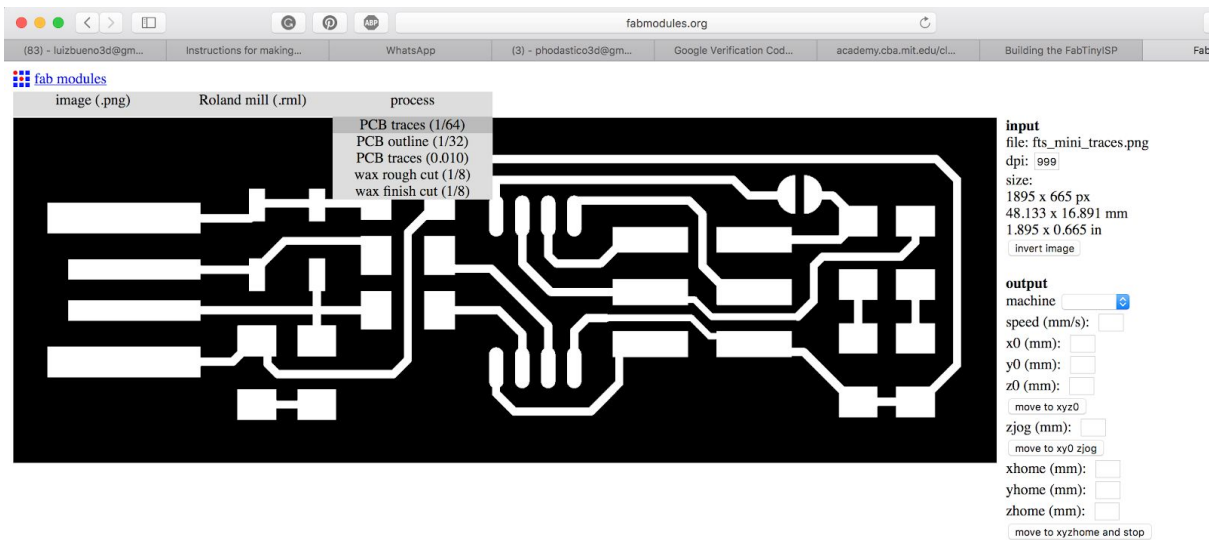
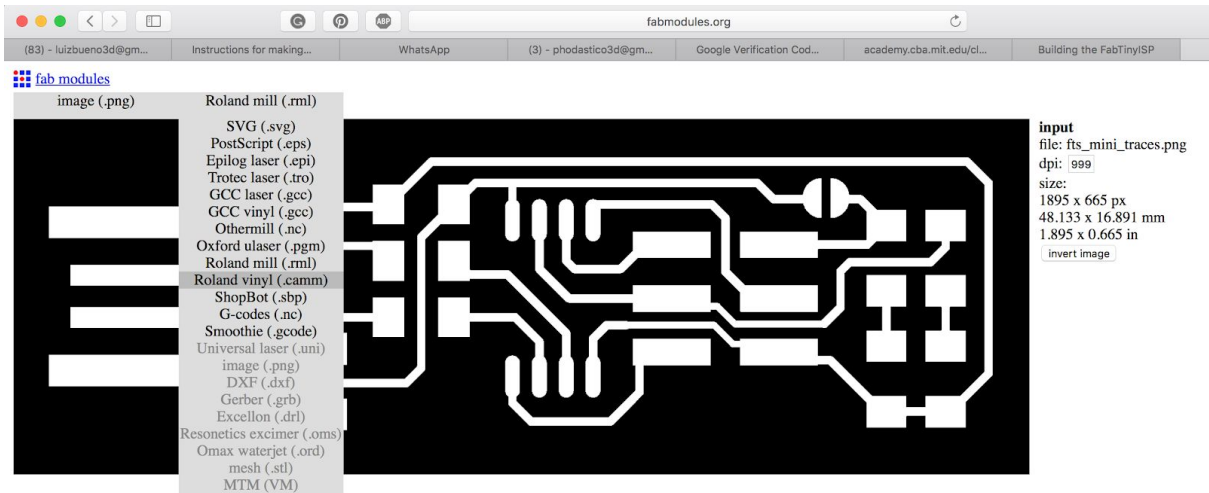
process?

image (.png) Roland mill (.rml) process



input
file: fts_mini_traces.png
dpi: 999
size:
1895 x 665 px
48.133 x 16.891 mm
1.895 x 0.665 in
 invert image

output
machine:
speed (mm/s):
x0 (mm):
y0 (mm):
z0 (mm):
 move to xyz0
zjog (mm):
 move to xy0 zjog
xhome (mm):
yhome (mm):
zhome (mm):
 move to xyzhome and stop



Note, if using a 0.3mm cutter for traces, the speed (output section) should be changed from 4mm/s to 3mm/s. This is because this is a smaller cutter and the machine needs to go slower!!!

image (.png) Roland mill (.rml) PCB traces (1/64)

input
file: fts_mini_traces.png
dpi: 999
size:
1895 x 665 px
48.133 x 16.891 mm
1.895 x 0.665 in
 invert image

output
machine: MDX-40
speed (mm/s): 4
x0 (mm): 0
y0 (mm): 0
z0 (mm): 0
 move to xyz0
zjog (mm): 2
 move to xy0 zjog
xhome (mm): 0
yhome (mm): 0
zhome (mm): 2
 move to xyzhome and stop

process

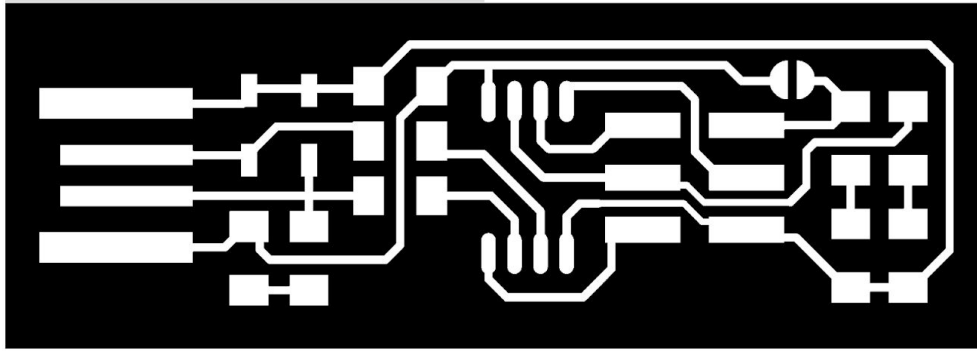
send command:

server:

direction:
 conventional climb
cut depth (mm):

tool diameter (mm):

Do note: zhome (mm): 2
(This number Always above 0)



input
file: fts_mini_traces.png
dpi: 999
size:
1895 x 665 px
48.133 x 16.891 mm
1.895 x 0.665 in
[invert image](#)

output
machine: MDX-40
speed (mm/s): 4
x0 (mm): 0
y0 (mm): 0
z0 (mm): 0
[move to xyz0](#)
zjog (mm): 2
[move to xy0 zjog](#)
xhome (mm): 0
yhome (mm): 0
zhome (mm): 2
[move to xyzhome and stop](#)

process
[calculate](#) [save](#) [send](#)
send command:
`mod_serial.py /dev/ttyUSB0`
server:
127.0.0.1:12345
direction:
conventional climb
cut depth (mm):
0.1
tool diameter (mm):
0.4
number of offsets (-1 to fill):
4
offset overlap (%):
50
path error (pixels):
1.1
image threshold (0-1):
.5
sort path:
sort merge diameter multiple:
1.5
sort order weight:
< 0: boundaries last
= 0: min distance
> 0: boundaries first
-1

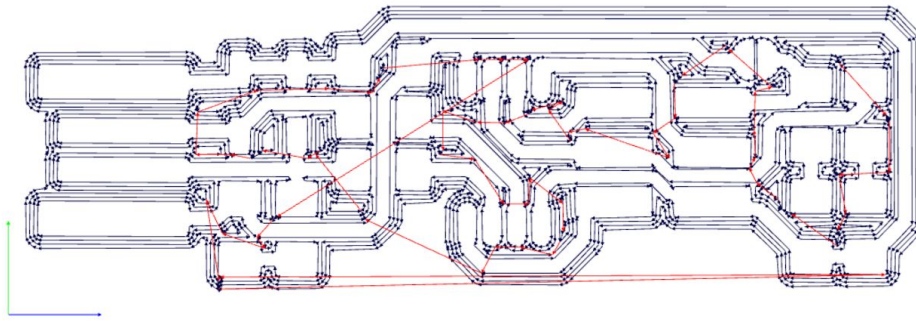
Number of Offsets: "4", if you've never soldered. "2", if you're comfortable.

left: pan, scroll: zoom, right: rotate, c: connections

image (.png)

Roland mill (.rml)

PCB traces (1/64)



input

file: fts_mini_traces.png

dpi: 999

size:

1895 x 665 px

48.133 x 16.891 mm

1.895 x 0.665 in

invert image

output

machine MDX-40

speed (mm/s): 4

x0 (mm): 0

y0 (mm): 0

z0 (mm): 0

zjog (mm): 2

xhome (mm): 0

yhome (mm): 0

zhome (mm): 2

process

send command:

mod_serial.py /dev/ttyUSB0

server:

127.0.0.1:12345

direction:

conventional climb

cut depth (mm):

0.1

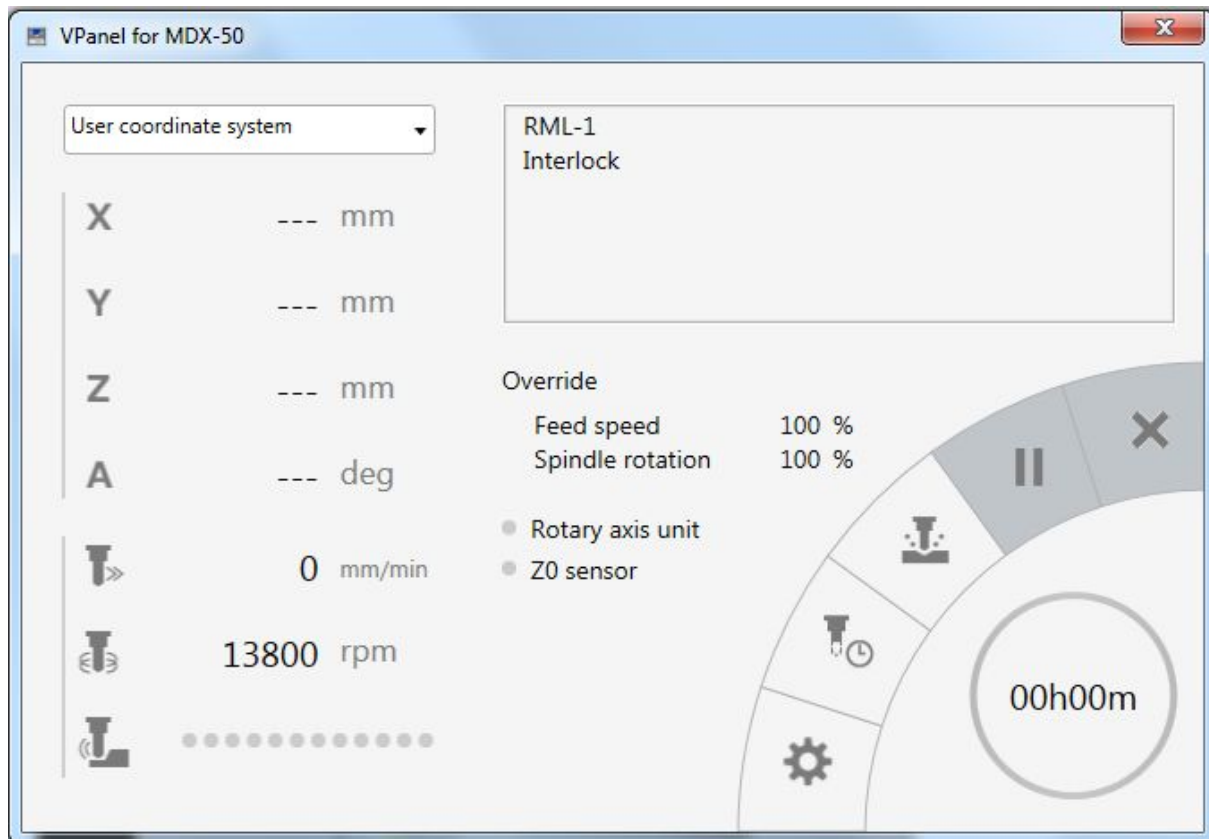
tool diameter (mm):

Once you hit Calculate, the image above should be displayed.

Save if you're satisfied with the tracing result.

By this point you should have an .RML file

Then open VPanel for MDX-50 software.



Verify if the machine is ready, the software will tell you.

Check if the XYZ are the same as the machine. (Eyeball it)

Loading your file: Click on the CUT button (middle button at 45deg). The job menu will appear, DELETE ALL of these, and click ADD, then select your .rml file as saved earlier.

Then click on OUTPUT to start the job.

Caution: Now it will start milling!

Make sure you check the machine is doing what you want EARLY ON! If there's a problem, lift the handle (best), or press the red emergency button (next best) to stop the machining.

Note: the OVERRIDE button can be used to change the speed (up or down) during the milling process.

4. Set up machine to cut out board outline

Select outline cutting tool.

Tool 1 on our machine:



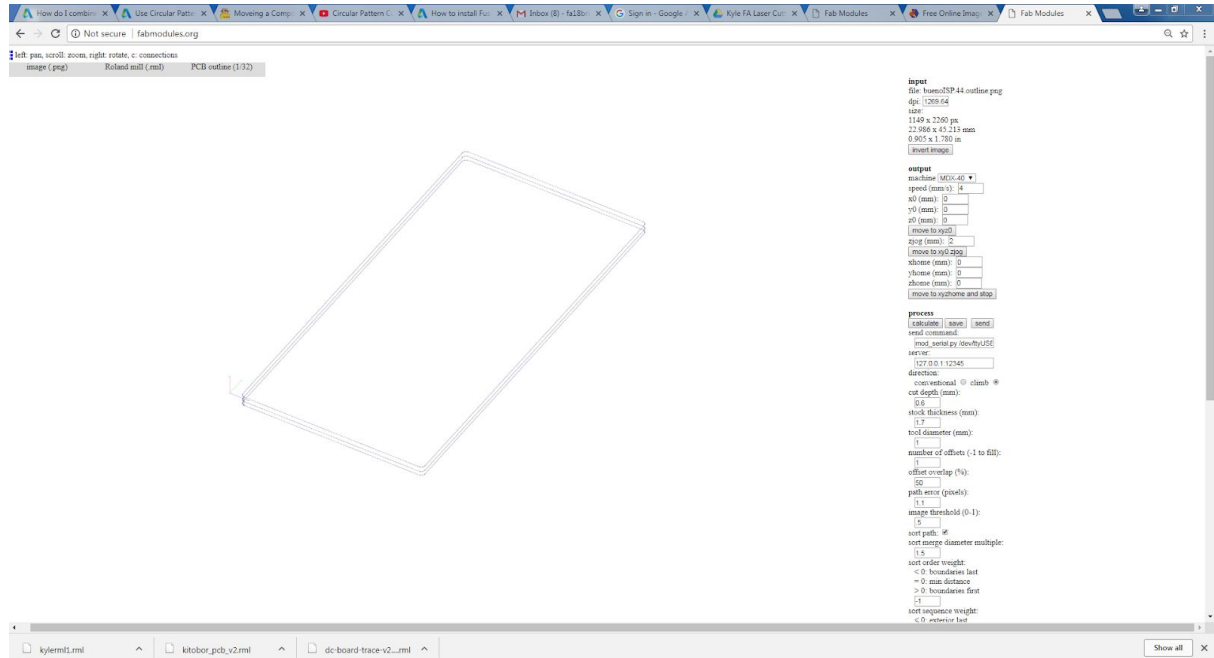
Send to User Origin

5. (FabModules) Prep file for cutting outline

Tool diameter: 1mm

Cut depth: 1.7mm (assuming material thickness of 1.6mm)

Tip: If you right click and drag, you can see a 3D version of the cut



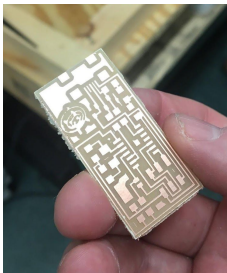
Follow same steps as loading <YOURNAME>traces.png but with <YOURNAME>outline.png into Vpanel: See [3. Prep file for cutting traces](#)

Success!

[Trace cutting](#)

[Board Cutting](#)

<Add image of each person's board>



6. Cleaning up the board

Sand down the rough edge of the board

Clean the top of the board with soap and thin wire-wool (pressure is important here, we don't want to remove traces)

Make it shiny :-D