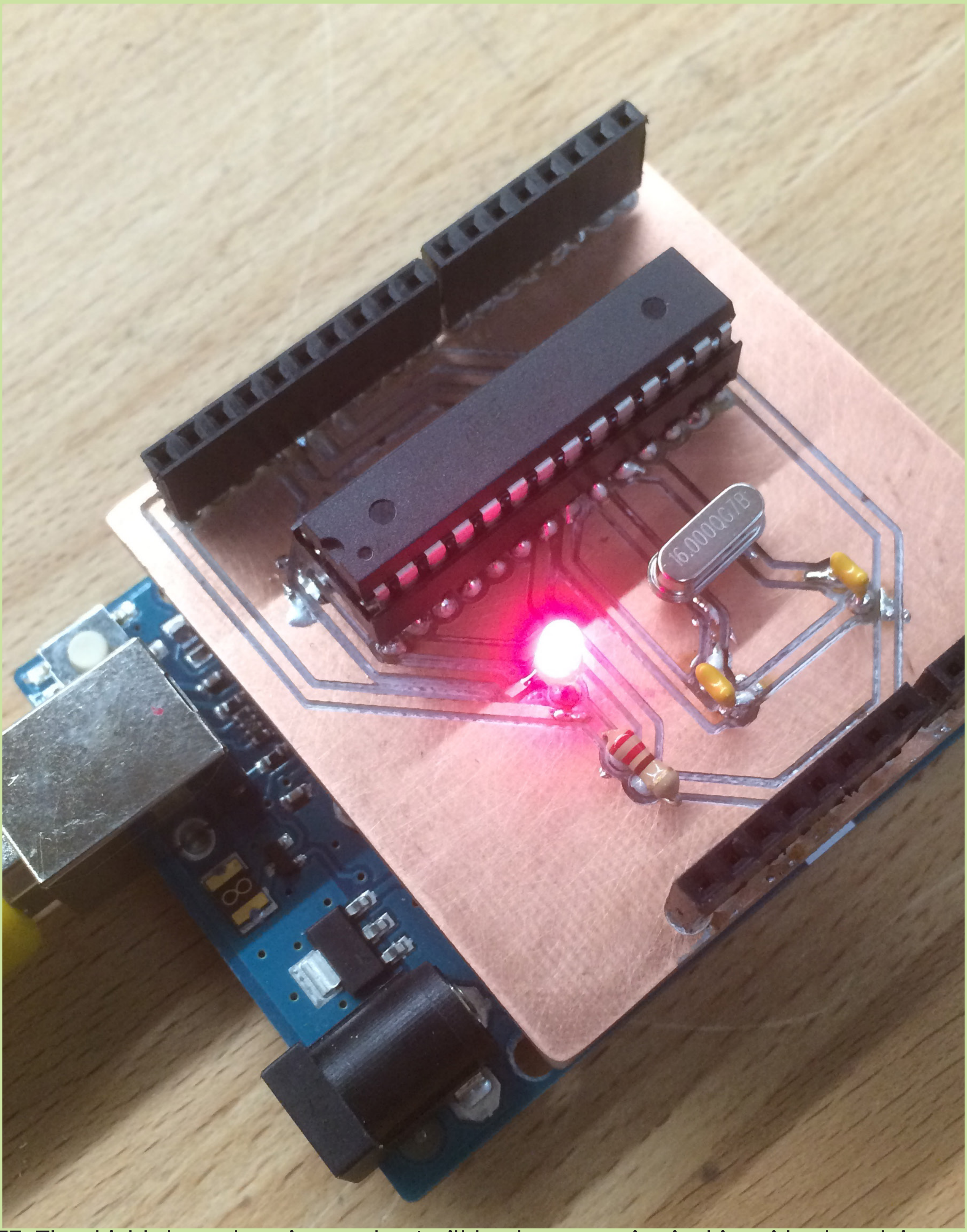
MILLING GUIDE

By Megan Bates

How to turn your arduino project into a beautiful, robust little shield. Ideal for exhibition pieces, projects you want to remove from your arduino without complete dissasembly, mass production modular pieces, or circuitry artworks.

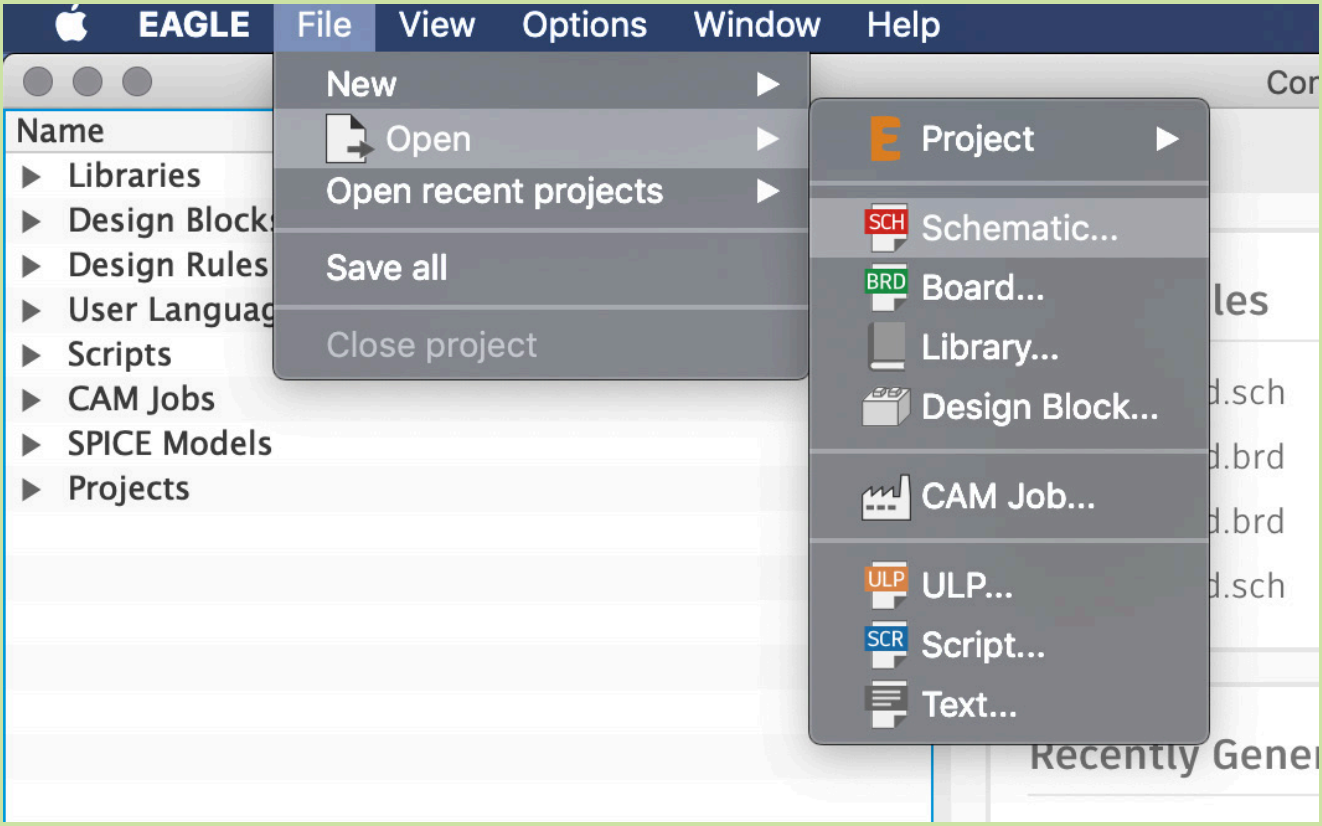


NOTE: The shield shown here is not what I will be demonstrating in this guide, though it would be very easy to adapt. This project can be found here: https://www.arduino.cc/en/Tutorial/ArduinoToBreadboard

STEP 1: CREATING YOUR CIRCUITRY DESIGN IN EAGLE

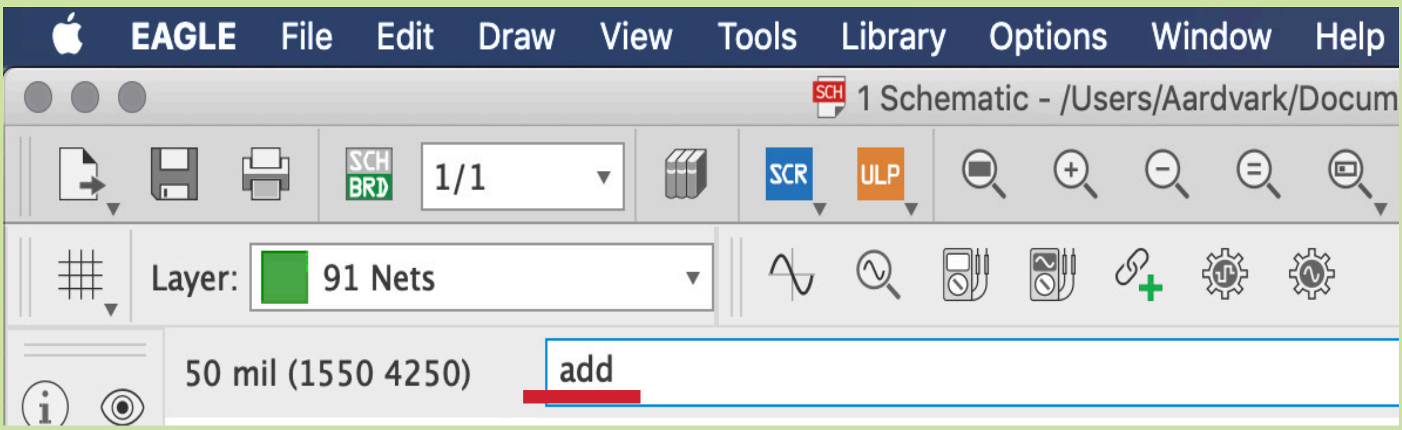
Download Eagle for free here: https://www.autodesk.com/products/eagle/free-download

On opening Eagle, create a new schematic:



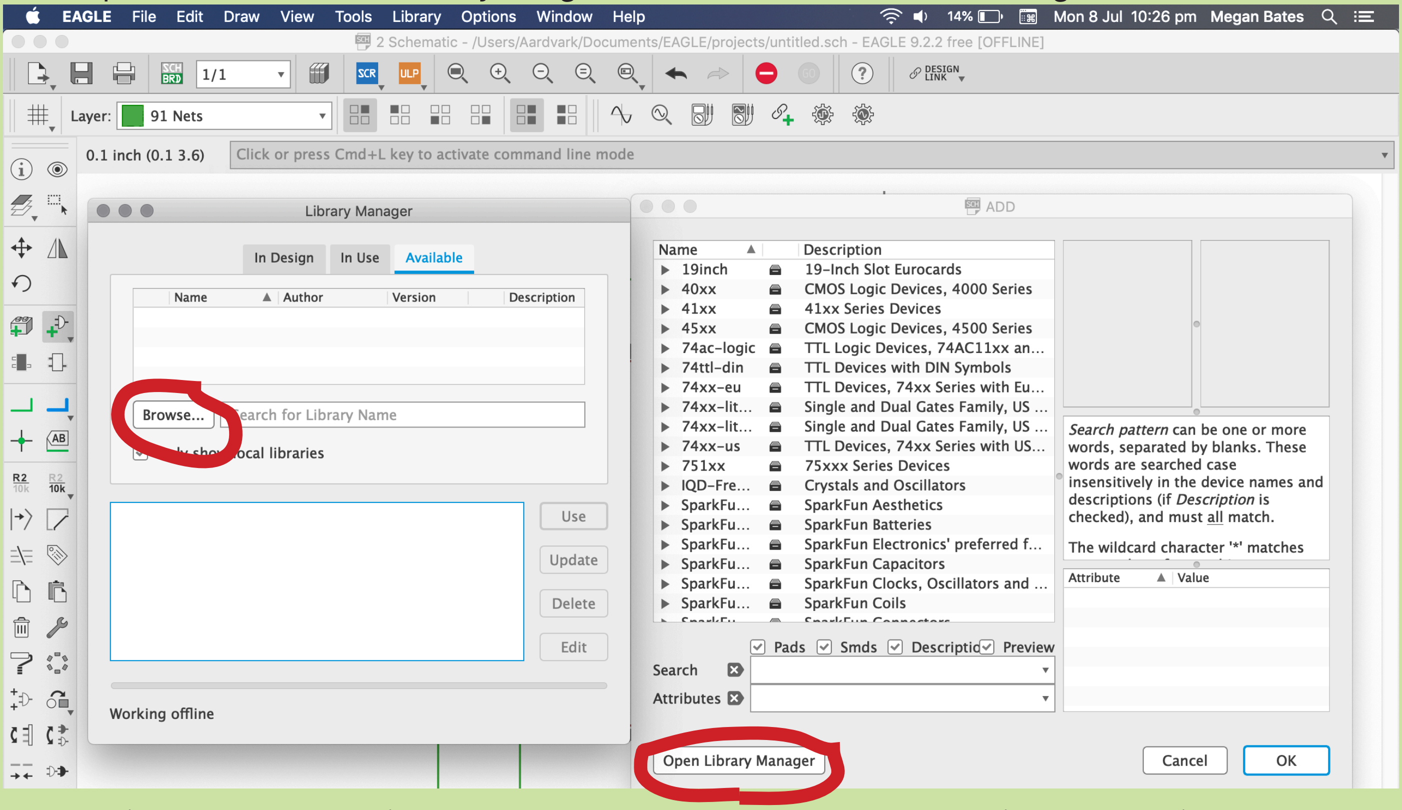
From here, the unusual part of eagle is using the text based controls. There is also a graphical control scheme, but some tools need to be accessed via the text control.

add: introduce a new type of schematic component into the Eagle schematic.
net: draw wires to connect your components
delete: remove components and wires
move: relocate components and wires
etc. They are fairly self explanatory and many of these have graphical equivalents in the toolbar.

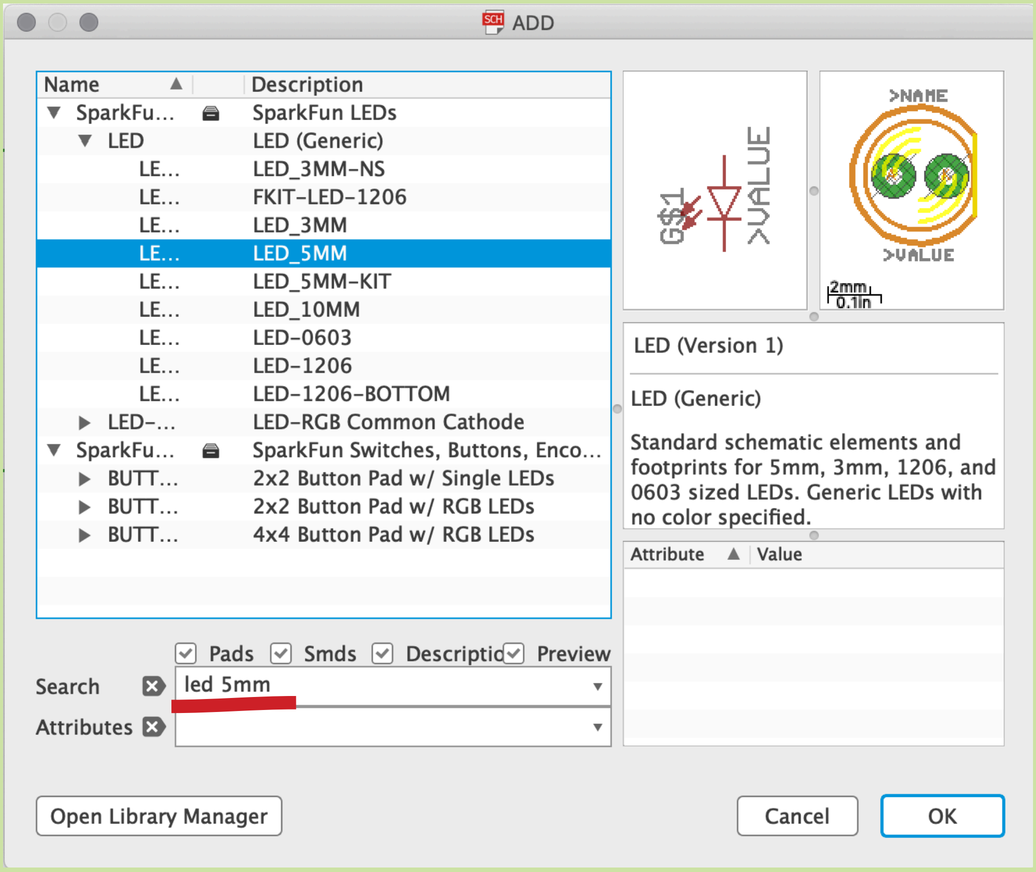


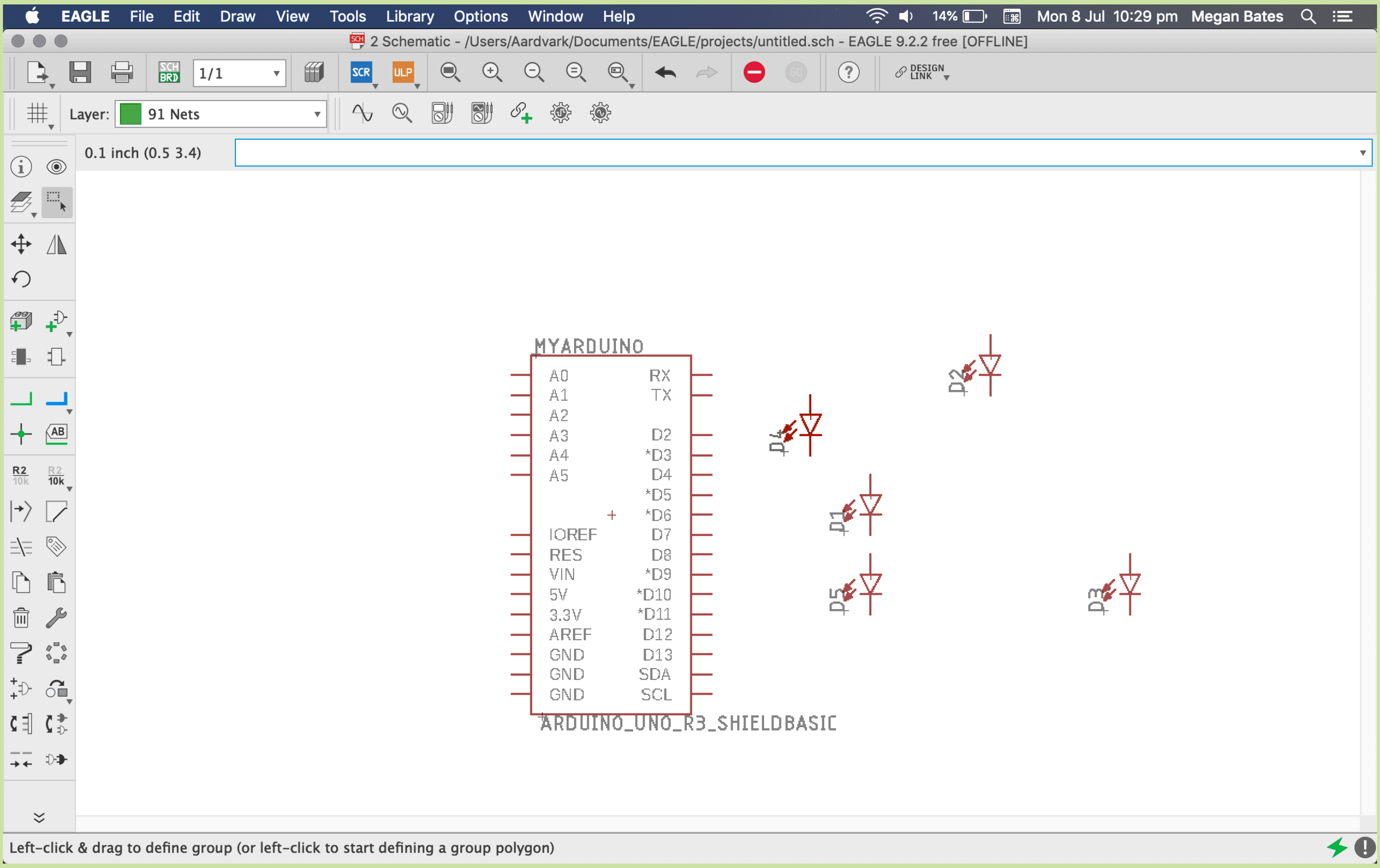
You need a library for Eagle to include the footprint information for Arduino. You can find an amazing collection of libraries for hobby electronics here: https://github.com/sparkfun/SparkFun-Eagle-Libraries but the only one which is completely necessary is the Sparkfun-Boards library.

You should be able to install your new library by opening the ADD window (by typing add into the schematic textbar), then opening the Eagle library manager. You can browse your computaer for the correct library using the 'available' tab and then clicking browse...



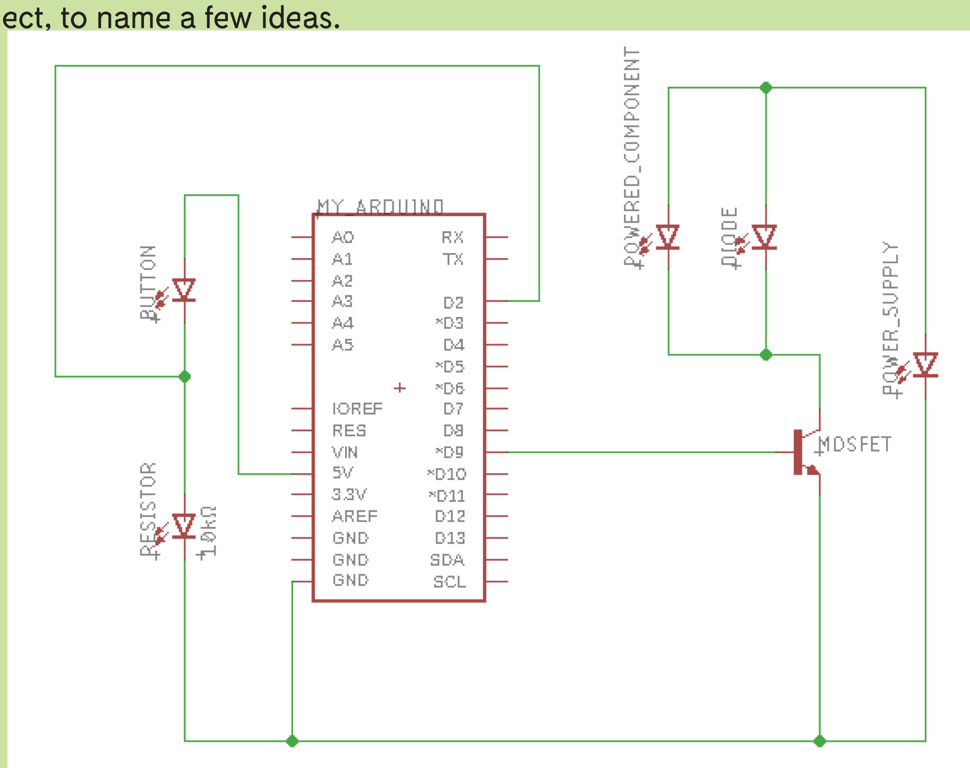
You will know your new library has been added when you can search in the ADD window for arduino, and the sparkfun arduino footprints appear. You will also want to include an LED footprint, for which sparkfun also has a good library.



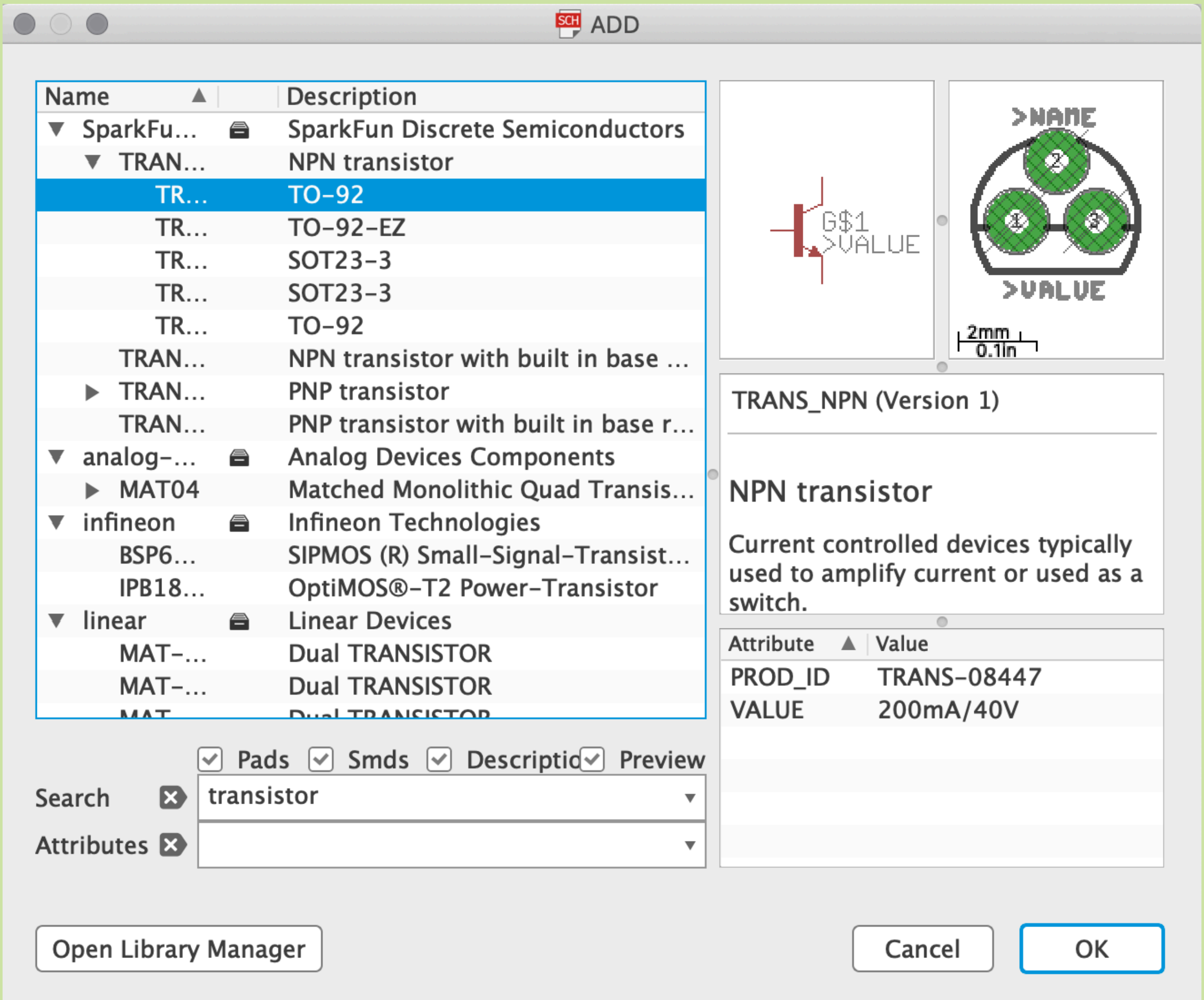


Your schematic window now may look something like this. You now have all the components you need to make a working schematic. I actually recommend using the 3mm LED footprint for almost any component you might want to add, as it will ensure your design is more consistent later. You can label components with the 'name' tool rather than including different kinds of components, and it streamlines things as you can just copy-paste an LED when you need to extend your schematic rather than navigating the ADD menu again.

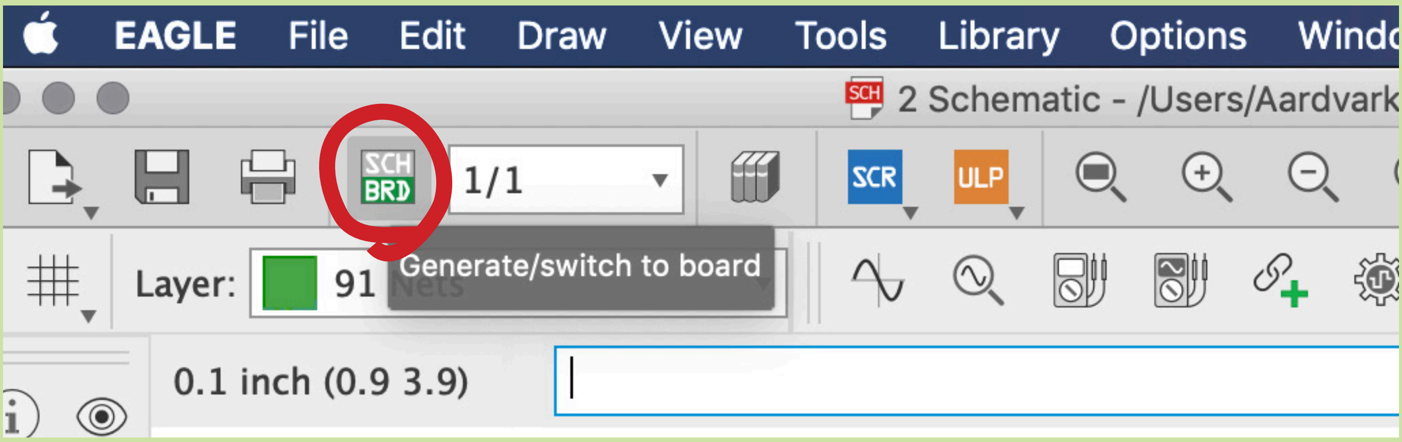
As an example, I am going to build in the following image a working schematic for an Arduino-controlled relay. The arduino will be able to control a power supply for a component which is much higher than the power supplied by the arduino. In the book, they use it to control a motor, but you could equally use a relay to control any high-power component, such as flickering a lamp on and off, controlling an electric lock on a door, or forcing a hard reset on another arduino project, to name a few ideas.

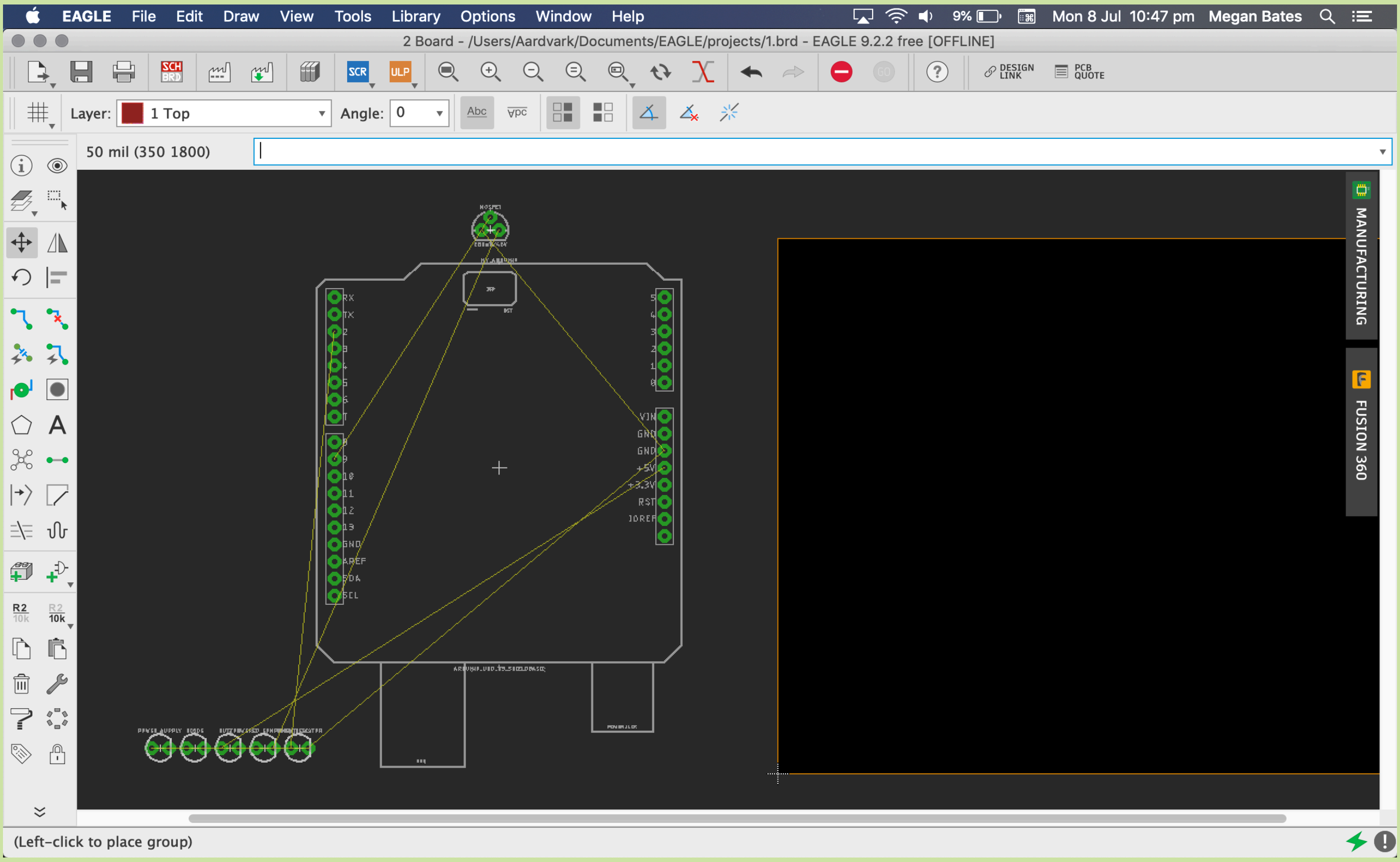


If you check this image against project 09: Motorized Pinwheel in the arduino book, you will see that this schematic is very nearly identical to the diagram they show. Creating a schematic can be very easy if you learn how to copy and adapt an example, so although this looks complex you don't need to be intimidated!
(You can find the project with schematic here if you don't have the arduino book) http://astenor.free.fr/Arduino/Projet_09/Project_09-EN-v2.pdf

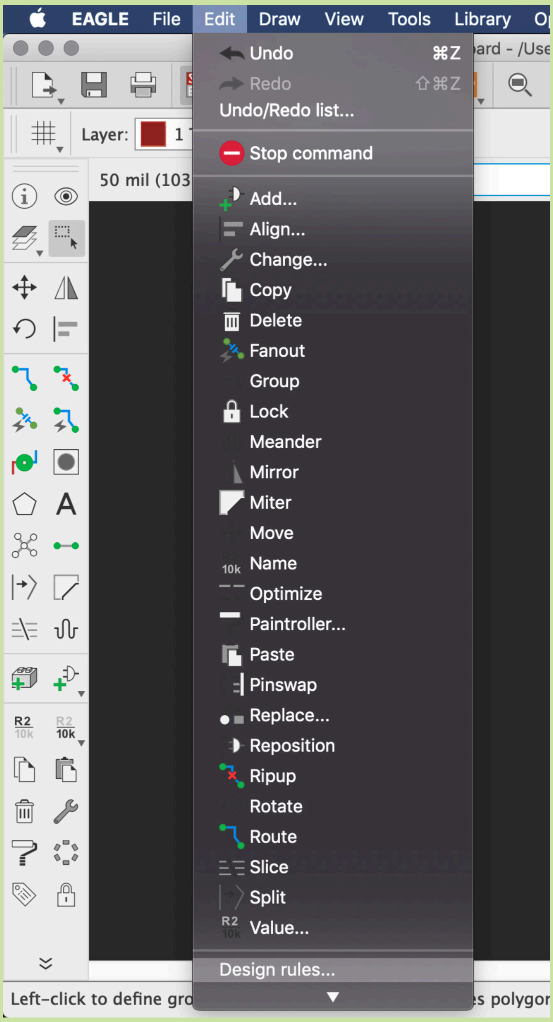


You may have noticed I have a component other than an LED in my scematic- I used this sparkfun diode footprint as shown above.
You are now ready to organise your board (based on your schematic). You can navigate here by pressing the switch to board button in Eagle.

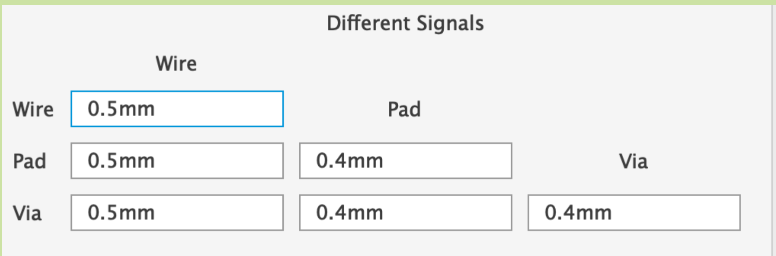




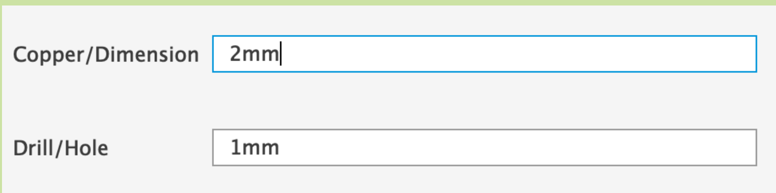
Your board window will now look something like this. You can arrange your components in the black area to allocate positions to your components. They will all be precisely scaled to fit perfectly onto your arduino once it is printed.



You now want to navigate to the design rules window. Here, in the clearance tab, adjust all spacings as follows (if you write mm the units will adjust automatically)



Then, in the distance tab, adjust settings as follows:

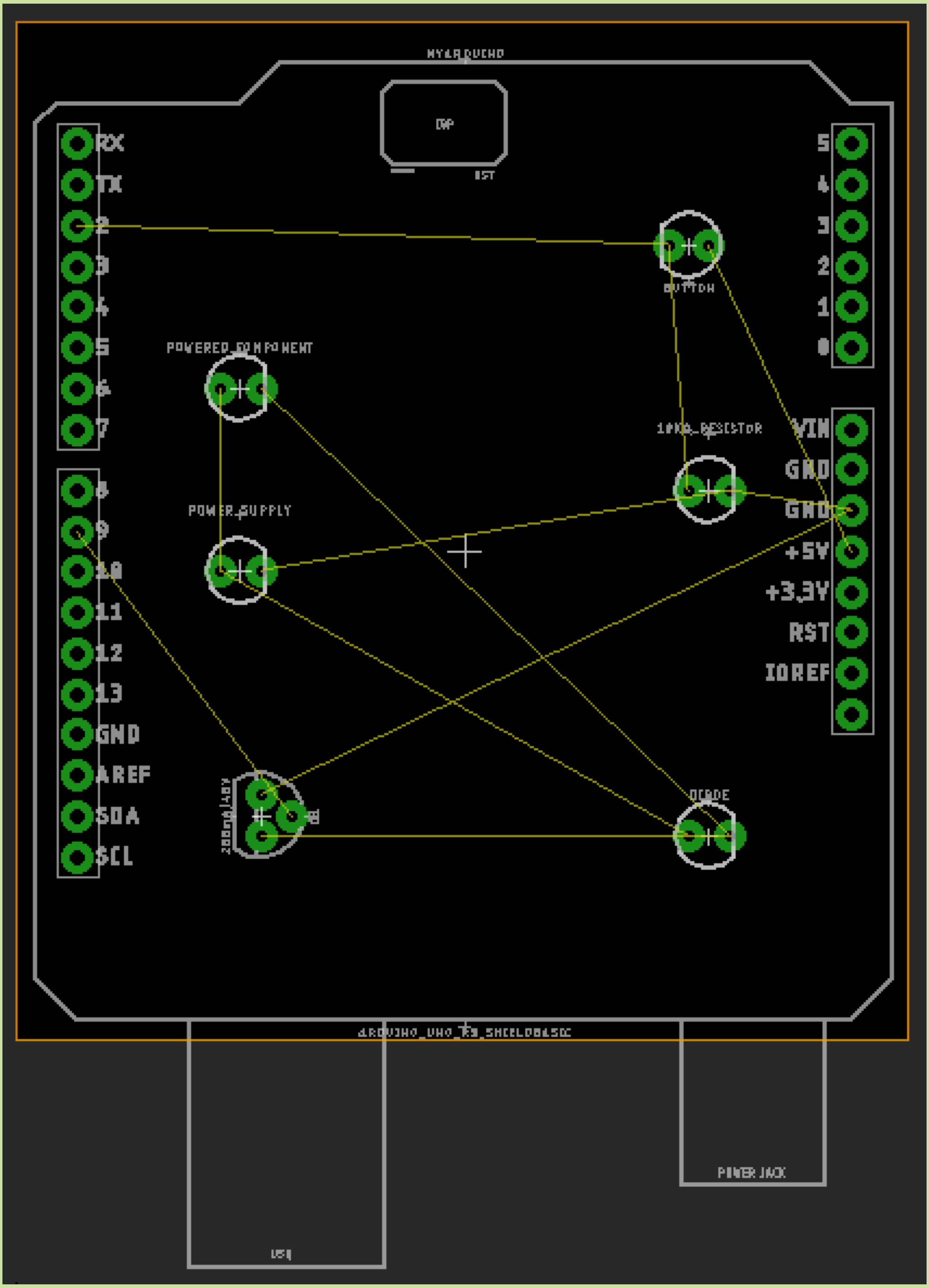


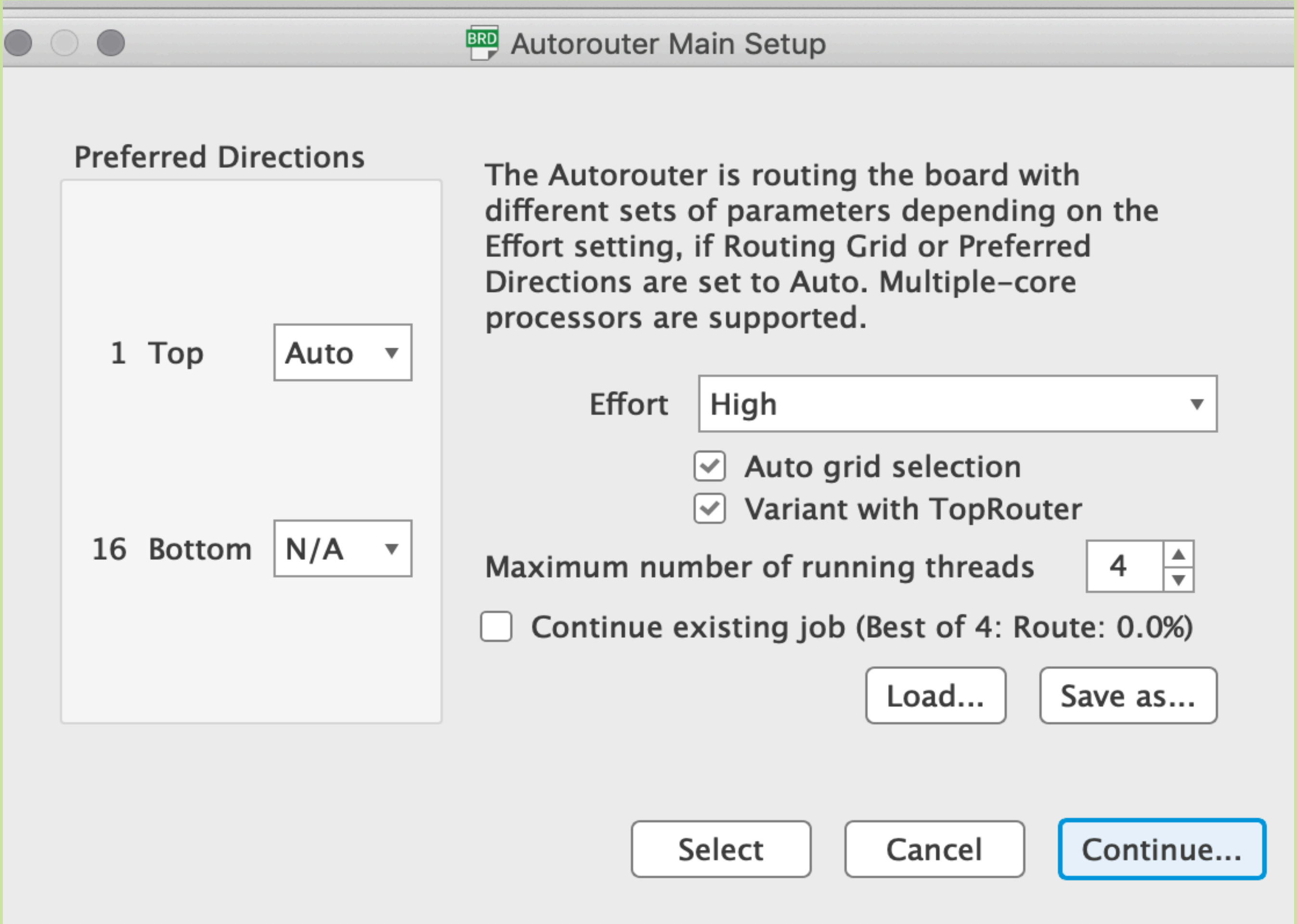
It is possible that later you may need to adjust these values for Eagle to be able to route your circuitry. Eagle will show you an image of what it means when you mouse over these text boxes, see if you can get a grasp of what they mean.

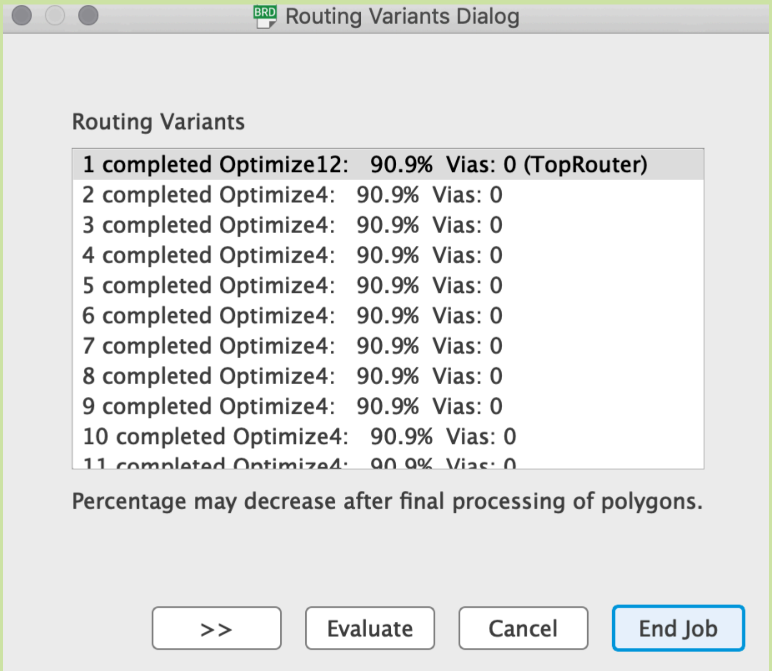
Minimum Width	0.5mm
Minimum Drill	0.5mm
Min. Micro Via	9.99mm
Min. Blind Via Ratio	0.5

Under the sizes tab, apply these values.

Roughly arrange your components, try to get the yellow lines arranged in a fashion which is reasonably uncomplicated to improve your chances of getting a good circuit design. You now want to open the autorouter window from Tools>Autorouter. If you set ‘auto’ for preferred direction, and ‘high’ for effort, you will get the maximum amount of computational effort for complex boards and be likely to get a 100% routing. Unless you want to create a double-sided circuit board (you probably don’t!) Set the ‘bottom’ preferred direction to N/A. If you change any settings, make sure to uncheck the ‘continue existing job’ tickbox.



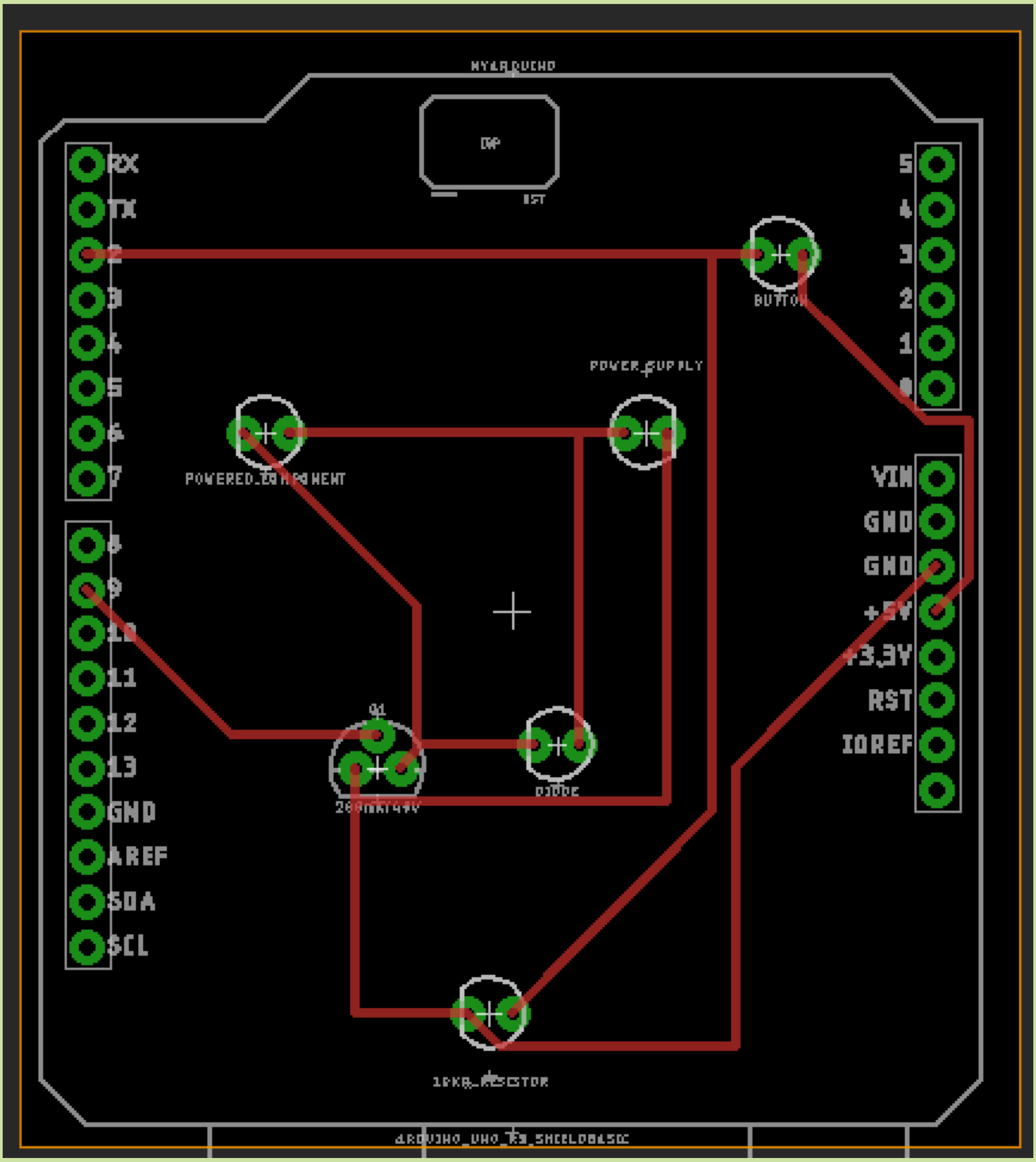




It is quite possible that on running the autorouter you will get outcomes which aren't 100% completed. If this happens, you can try adjusting the design rules (though just remember closer wires increase the chances of problems on the mill), rearranging components, or increasing the size of clearance of the black board area around your components. Don't forget, if you change any settings, make sure to uncheck the 'continue existing job' tickbox on rerunning the autorouter.

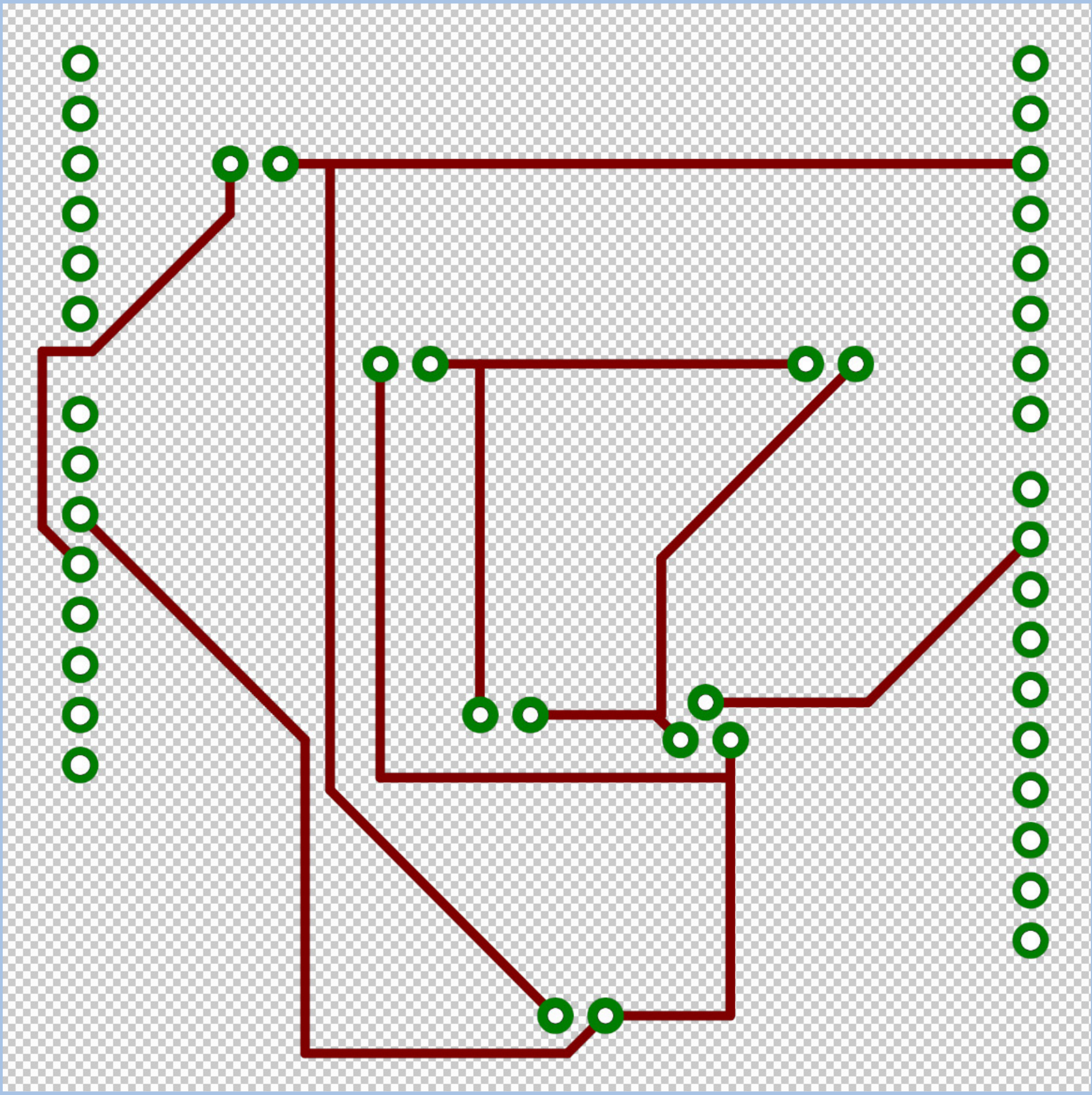
You can evaluate to see which connections haven't been made, and move components around to increase your chances of a good autoroute. If you do this, you then need to remove the routes before trying again. select everything with cmd+A, pick the 'ripup' tool, right click on a trace and select 'Group:Ripup' > 'Trace, via, or polygon'

Having rearranged the components a little, I pick the autorouter solution which I think looks best. I now need to export the design. To do this, I go to the View > Layer Settings window. I then turn off all layers using the 'hide layers' button, and switch on only the 'top' and 'pads' layers. I now export my board design using 'File' > 'Print'. I make sure the settings are 'print to file' and 'actual sclaing 1.00'. We are now ready to move onto tweaking our design in photoshop!



STEP 2: TWEAKING YOUR CIRCUITRY DESIGN IN PHOTOSHOP

Open your exported PDF into Photoshop (or GIMP etc). Check as you import, the resolution should be 800 pixels per inch. I recommend cropping your circuitboard to have a little bit of breathing room around the design.



Your goal at this point is to produce 3 png images, examples of which I will show on the next page. I recommend setting colours to pure black and pure white, which you can streamline by using Image > Adjustments > Black & White. I also recommend using a 30 pixel wide pencil tool for drawing holes as it will make your life easier later.

If you want, you can use this point as an opprtunity to stylise the board, adding design work or making the pathing smoother if you like. My examples will be straightforward.

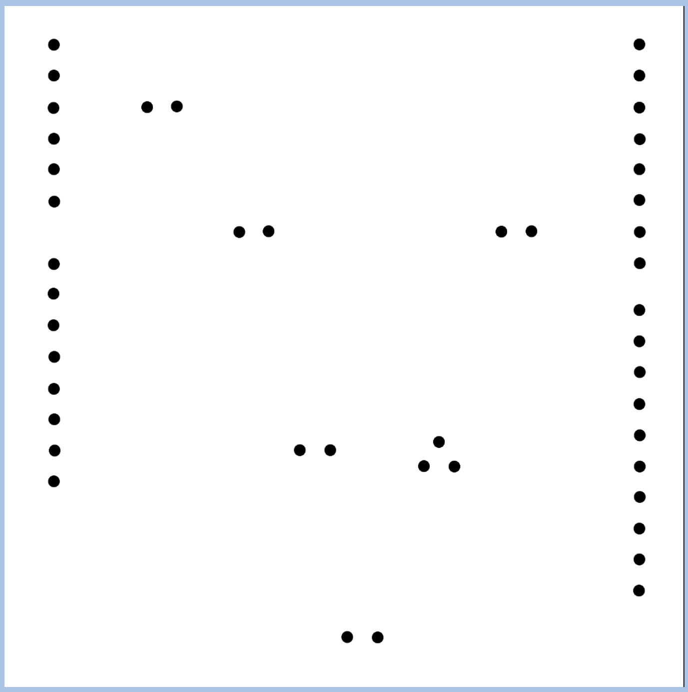


Image 1: draw a 30pixel wide (ideally using an unaliased pencil) dot over each center of a pad, where we will drill a hole to push our components into the board

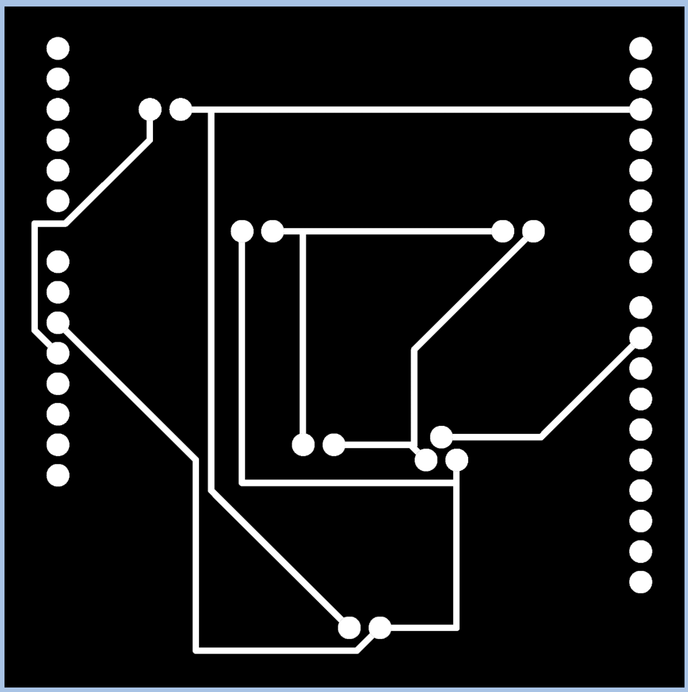


Image 2: The traces from Eagle, with holes in the centre of pads filled in. Make sure this image is pure black and pure white with a continguous paintbucket. It is worth scanning for any stray pixels while you are working on this- even small areas of the wrong colour can show up in the trace design later.
If you want to easily invert your image to be white on black, use Image > Adjustments > Invert.

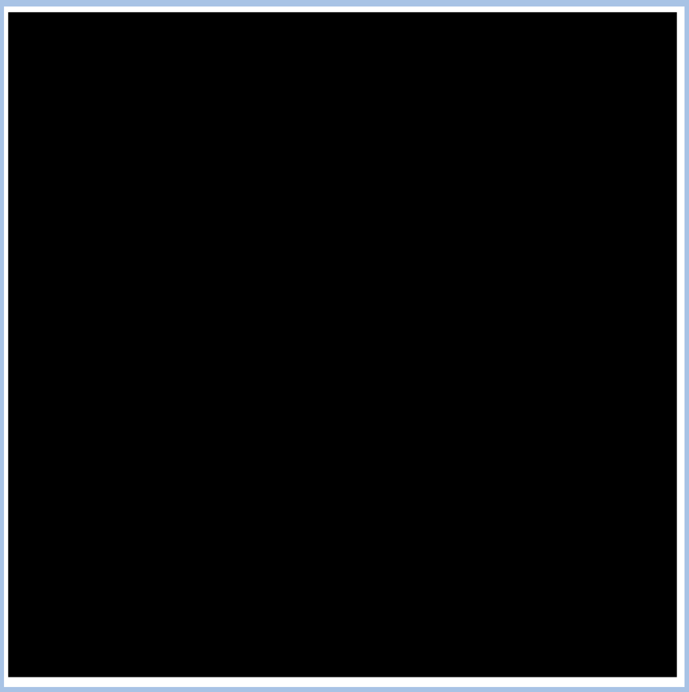


Image 3: The outline of your board. This can be a fun shape if you like, but I have used a straightforward square.

Make sure to save each of these files onto a memory stick as a png file.

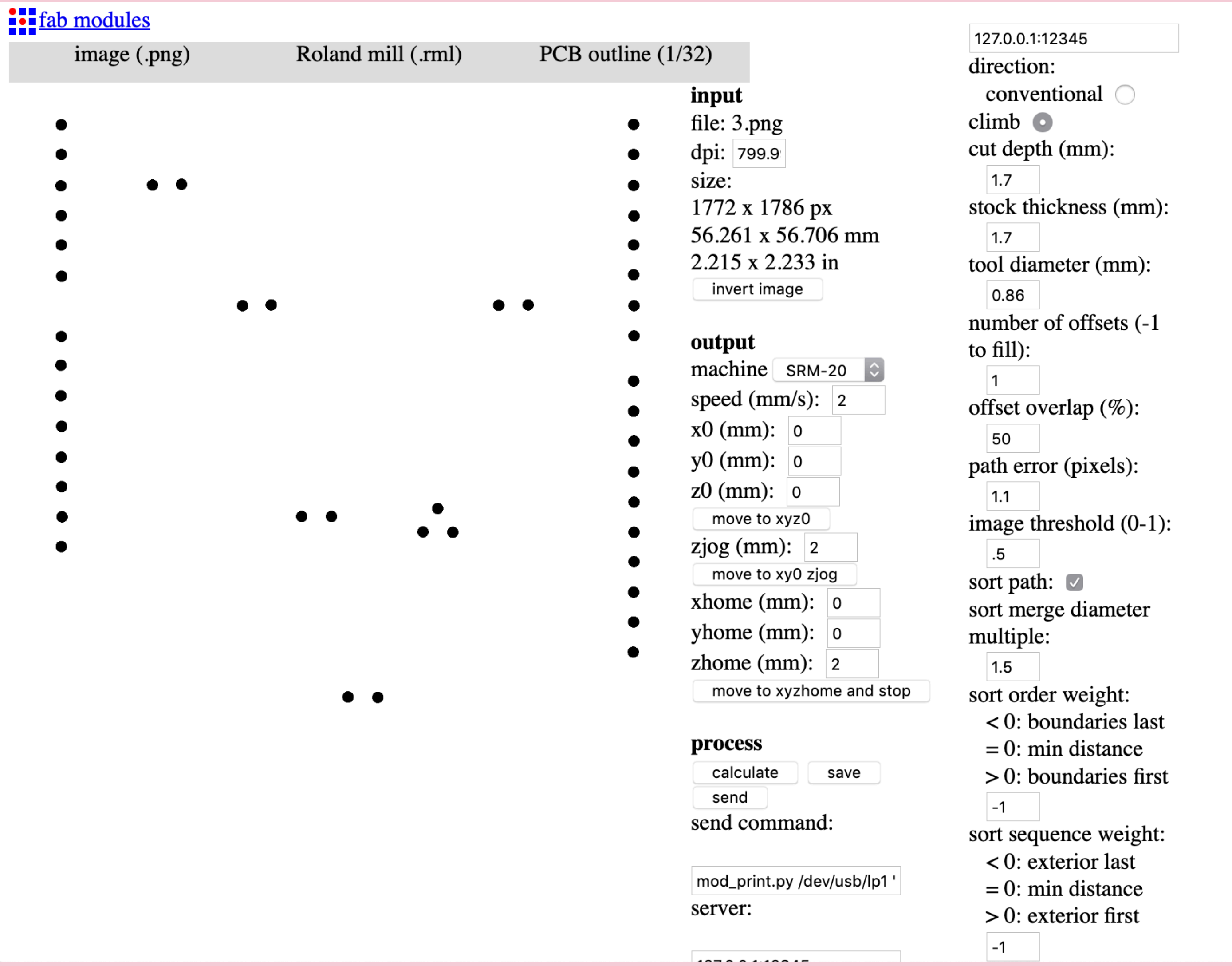
You are now ready to move on to building our milling files in fabmodules.

STEP 3: CREATING YOUR MILLING FILES IN FABMODULES

I recommend bringing your png files onto the Roland Mill computer via a memory stick at this point, and building your milling files on the hatchlabs mac. Open a web browser and navigate to fabmodules.org within the virtual machine.

If you are having problems accessing your memory stick files from within the virtual machine, try ejecting it from within the mac operating system and then accessing the USB within Windows. You will need to connect to the Roland Mill as well once you turn it on.

Now is a good time to check your scaling. There are callipers in the hatchlab. Check that the dimensions from fabmodules match the size you want your PCB to be.



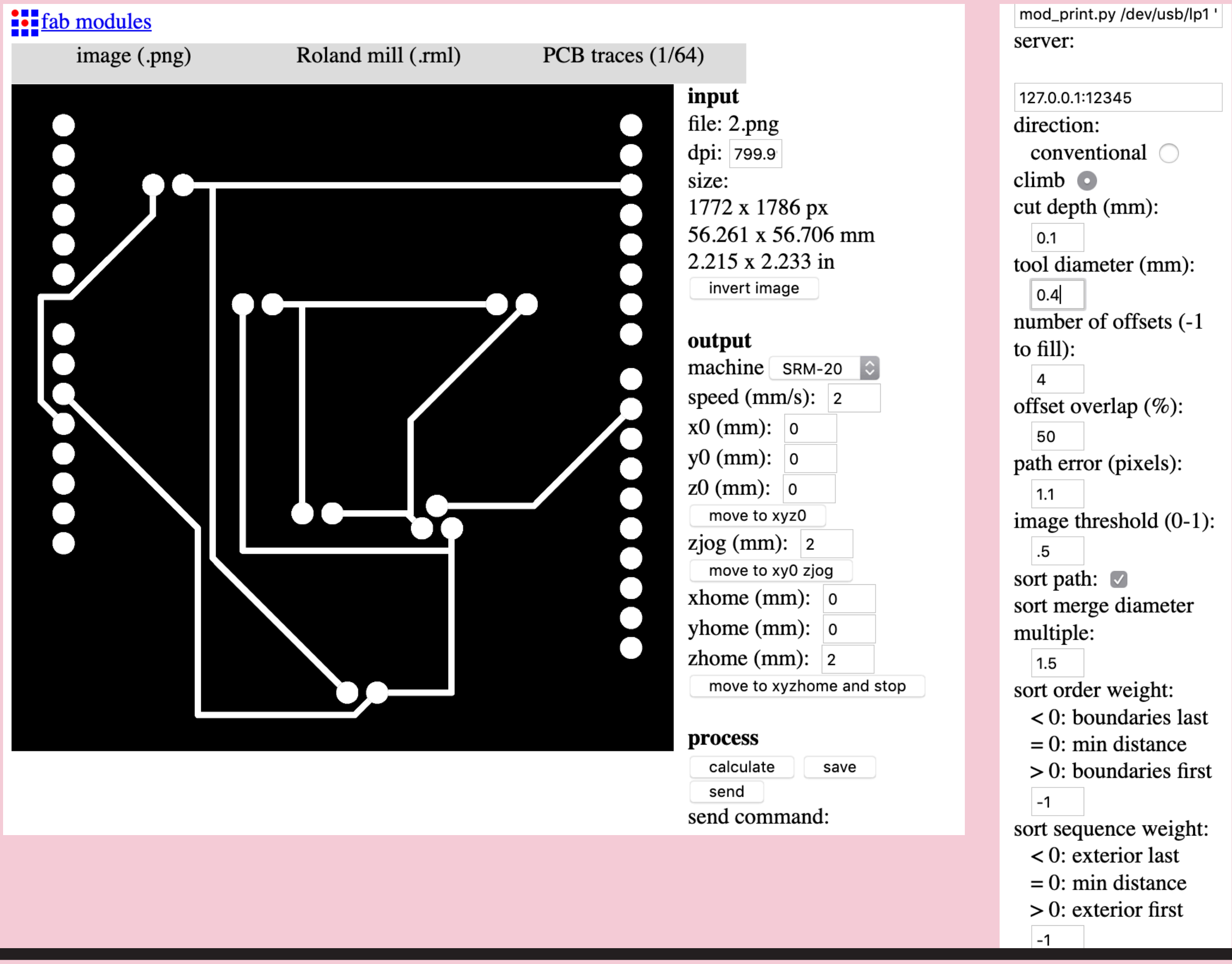
If you copy the settings as shown here, you should have no problems milling a design which is constructed as described in the rest of this guide. That said, there are some settings to pay extra attention to.

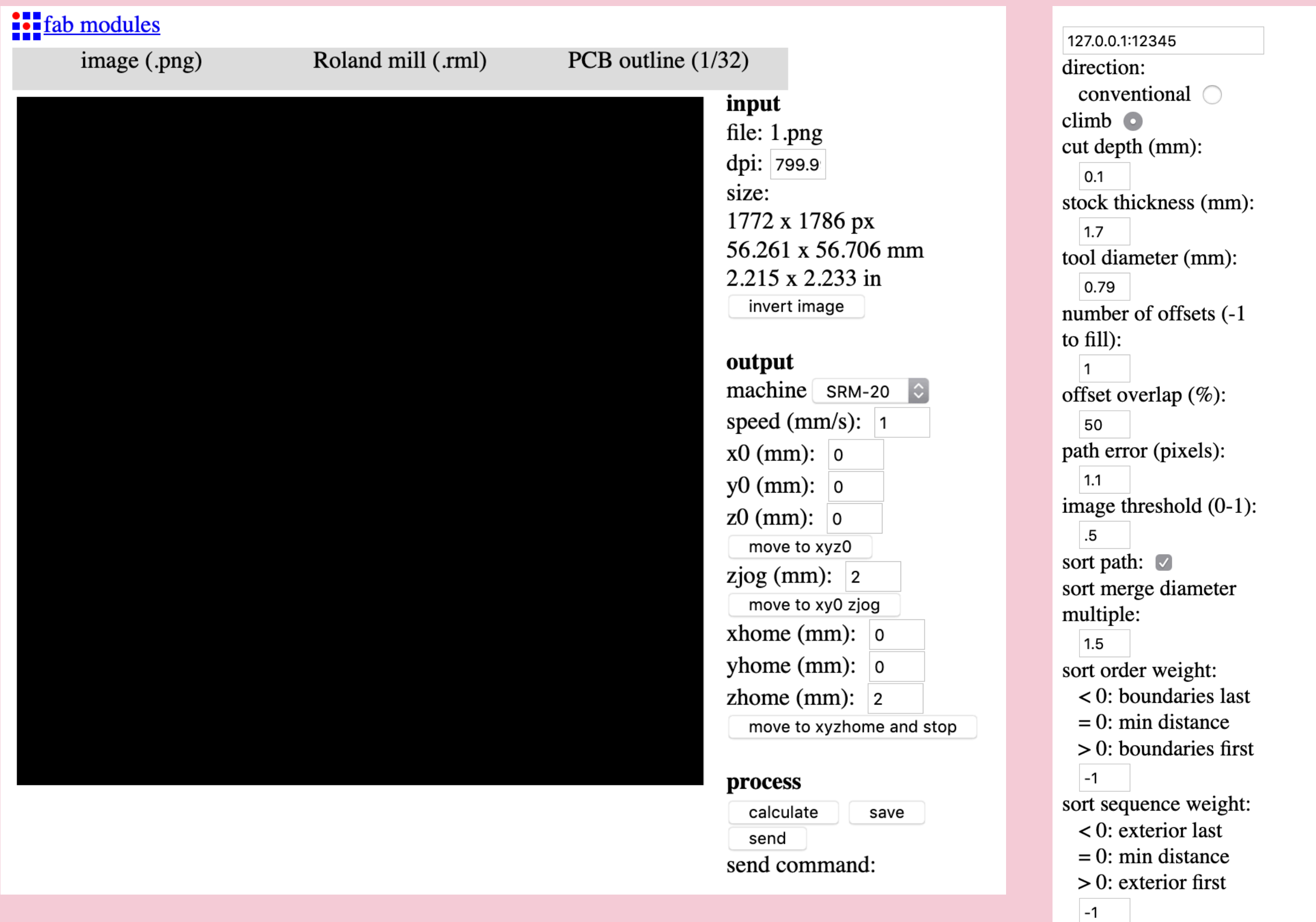
Generally: Make sure the file type is .rml. The 'process' will depend on the image, check that selection is correct. always ensure zjog and zhome are more than 0, or the drillbit will crash into the surface. Don't mess with the 'send command' or 'server' boxes from how they are autofilled. You shouldn't need to adjust any boxes after the 'offset overlap (%)' one.

Holes image: Make sure both the stock thickness and stock depth are a little greater than the thickness of your board. Make sure the tool diameter is 0.86, I found this worked with 30pixel wide circles.

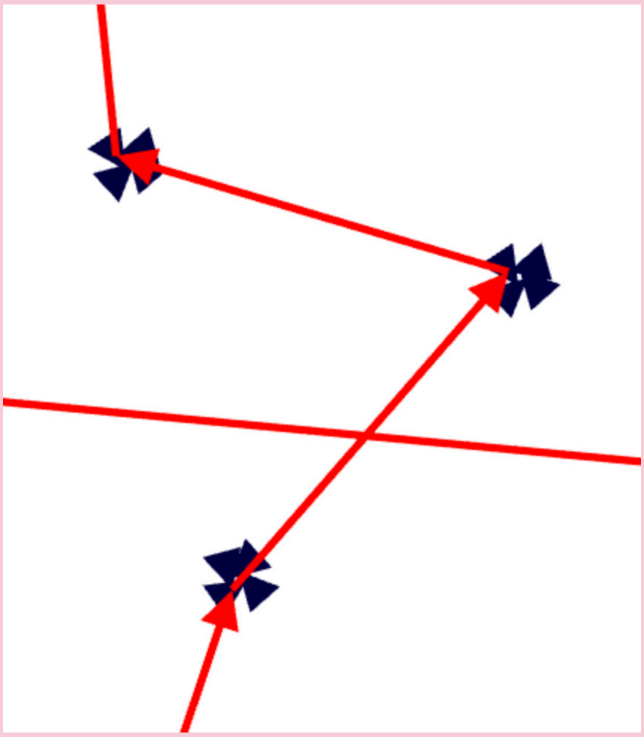
Trace image: You may want to increase the 'cut depth' value if the mill is not scoring well. Feel free to experiment with the number of offsets to change the look and speed of how your board is cut. You can preview this pressing the 'calculate' button.

Outline image: Keep your cut depth and speed very low, this is the stage which will break drillbits.

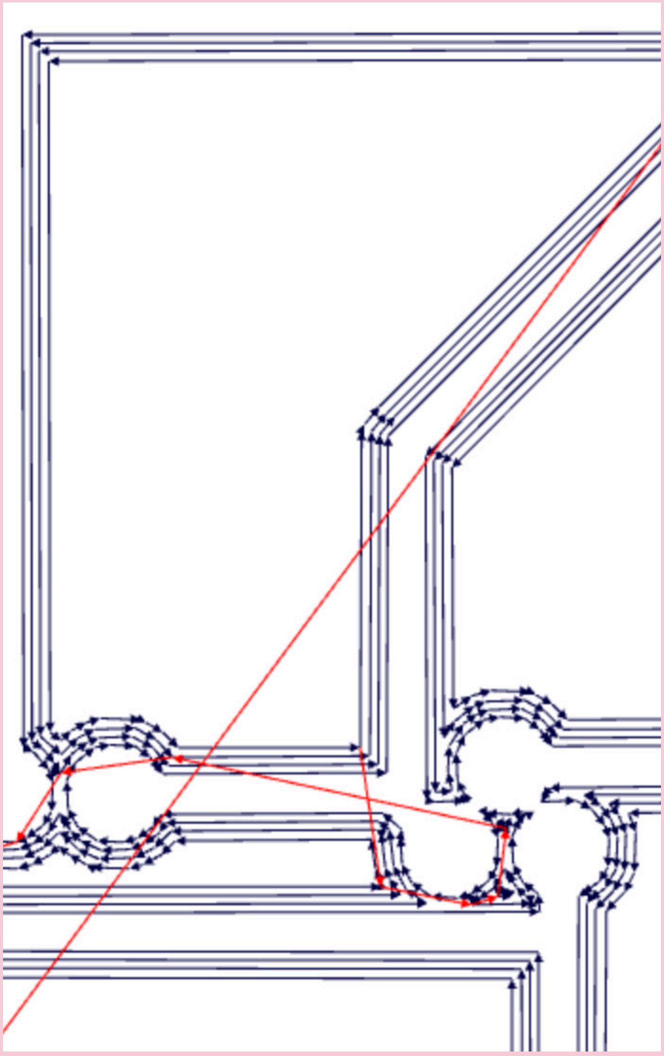

fab modules
image (.png)
Roland mill (.rml)
PCB traces (1/64)
input
file: 2.png
dpi: 799.9
size:
1772 x 1786 px
56.261 x 56.706 mm
2.215 x 2.233 in
invert image
output
machine SRM-20
speed (mm/s): 2
x0 (mm): 0
y0 (mm): 0
z0 (mm): 0
move to xyz0
zjog (mm): 2
move to xy0 zjog
xhome (mm): 0
yhome (mm): 0
zhome (mm): 2
move to xyzhome and stop
process
calculate
save
send
send command:
mod_print.py /dev/usb/lp1 '
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
sort sequence weight:
< 0: exterior last
= 0: min distance
> 0: exterior first
-1


fab modules
image (.png)
Roland mill (.rml)
PCB outline (1/32)
input
file: 1.png
dpi: 799.9
size:
1772 x 1786 px
56.261 x 56.706 mm
2.215 x 2.233 in
invert image
output
machine SRM-20
speed (mm/s): 1
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:
127.0.0.1:12345
direction:
conventional
climb
cut depth (mm):
0.1
stock thickness (mm):
1.7
tool diameter (mm):
0.79
number of offsets (-1 to fill):
1
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
sort sequence weight:
< 0: exterior last
= 0: min distance
> 0: exterior first
-1

I really recommend always pressing the ‘calculate’ button before saving the files, especially for drilling holes and drawing traces, so you can check the file is processing properly. Keep an eye out for strange features you weren’t expecting– sometimes a stray miscoloured pixel can break things. You can try adjusting the ‘image threshold’ value if you run into problems.



On pressing calculate, when you zoom in on your holes file, it should look like this. Not drawing a circle, but with arrow vectors in extremely localised points. If you see a circle appear, adjust the ’tool diameter’ to increase until the calculation is drawing nothing, and then reduce the value again util you get a result similar to this one. Tracing a circle will break the drillbit, so finding the right value is worth your time.



On pressing calculate, when you zoom in on your trace file, it should look like this, a good number of vectors drawn around each trace, and a generous amount of room for both traces and pads.

It is quite possible that like in this example, you will have pads which are connected. This is preferable to having the machine remove too much material, but you will need to adjust the board once it is printed and use a knife or file to remove the small amount of material connecting the pads in order for your board to work properly. If, like with this example, the component you are attaching isn’t a specific one, you can manually adjust the positions of the pads in photoshop to allow for more room and have the mill clear the pads properly.

I would recommend this if you wanted to mass produce a board, but usually I would remove areas like this by hand, since it is a good idea to go over the traces with a file after milling anyway.

We are now ready to move onto milling proper!

STEP 4: USING THE ROLAND MILL

An example of the kind of material you want to use for milling: https://uk.rs-online.com/web/p/plain-copper-ink-resist-boards/2192117/
make sure the size is big enough for your project!

APPROACHING PROCESSING YOUR PCB
I recommend for your first time:

1) testing hole drilling
2) testing trace cutting
3) testing board outline cutting
4) drilling holes
5) drawing traces
6) cutting the board outline
7) Hand filing away stray bits of metal

This will require changing drill bits many times, but it is a good idea to cut the board after drawing traces so it will be well attached when it's drawing traces. Drilling holes is the fastest stage of cutting, so I recommend doing this first to check if your design is the right size etc. before you spend ages waiting for traces to be cut.

BEFORE YOU START CUTTING
Check that the bed is level. Firmly attach the PCB board to the bed with double sided sticky tape. It should not move no matter how hard you try to remove it with your fingers. Make sure there are no obstacles the machinery could collide with. Use the mini Dyson hoover to clear out the inside.

NOTES ON EACH STAGE OF MILLING:

<u>Holes:</u> Use a thin drillbit- eg. 0.8mm. This stage will be pretty quick, and will show you any glaring mistakes (eg. your design being too big for your board) straight away, which is helpful. Check you are drilling all the way through before you move your board!

<u>Traces:</u> Use a V-shaped carving bit. Pause your trace cut early to check the cut is deep enough. You should be cutting through the copper, but not getting rough edges on your cut. If the cut is too shallow, feel free to re-zero the z position slightly lower than before (you can do this off the edge of the copperboard) or adjust the cut depth in fabmodules.

<u>Outline:</u> Use a thin drillbit- eg. 0.8mm. This is where your drillbit will break! If you have time, run this as slow and shallow as you can. If you need to go faster, cut the first couple of outlines and then speed things up once you have a rut cut for the drill to trace in.

ZEROING THE MILLING MACHINE

Open the VPanel program in the virtual machine.

Choose User Coordinate system from the dropdown box.

Zeroing: bring the tip as close as possible using continuous, then x100, then x10 cursor step if at all close to the board. Undo screw until it drops (on the UPPER alan key screw hole), tighten it again. Tell VPanel that the tip is zeroed. Don't try to calibrate by eye. Zero the Z axis only, you will get more consistent zero-positions this way. Don't zero X-Y because the machine will lose track of where your board is between drill changes.

If you have just loaded a sheet to mill, DO zero X-Y, at the closest-left corner of the sheet. If you are starting a new board on a sheet which has already been cut, find the bottom-left corner of a clear area. measure to see if your board will fit against the fabmodules sizing dimensions.

VPANEL

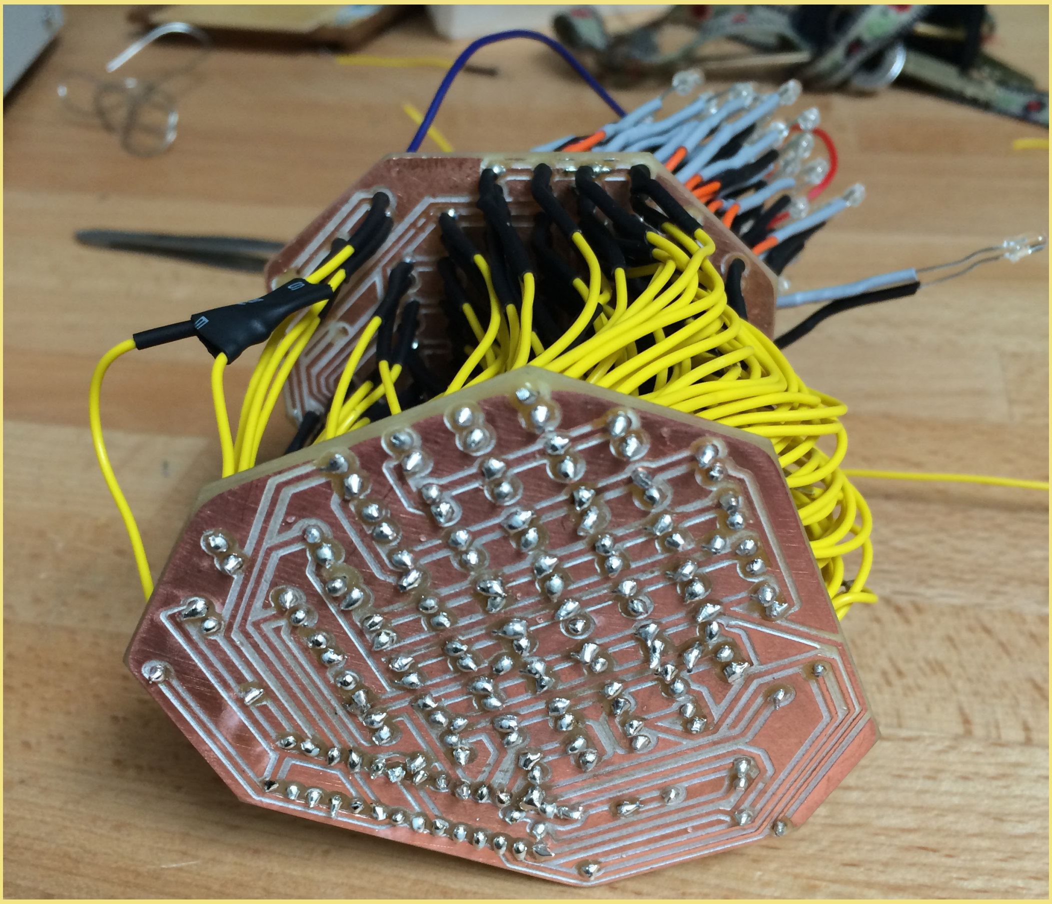
Press 'view' to look at your work. Wait until its stopped moving to open the mill door. You can resume your trace- the mill will remember the X and Y locations, so it can continue where it left off.

Press CUT to load up your .rml file.

! DELETE EVERYTHING ELSE IN THE LIST EVERY TIME YOU LOAD UP A NEW FILE. THE MILL WILL QUEUE OTHER THINGS IN THE LIST BEFORE YOUR WORK AND CUT THE WRONG FILE !

When you press Output the mill will start cutting. Keep an eye on what its doing and press CANCEL if the machine is cutting the wrong file, or something else is going wrong.

Projects I have completed using the Roland Mill-



If you have any further questions, feel free to contact me at meganlaurabates@gmail.com.