I recently just ran my first turret board on my 3018 CNC router. Long story short: It was a pain to figure out for someone with little to no experience with CNC work. I decided to make this guide so that maybe others will have an easier time than I did (or, let's be honest, so I can remember what I did the next time I try to create a turret board). I'll try to cover the nuts and bolts on how to easily make your own board, plus maybe a tip or two that's related.

First things first: You'll need the software. I use the following:

For my layouts, I use DIY Layout Creator: https://bancika.github.io/diy-layout-creator/

To generate G-code, I use Inkscape: https://inkscape.org/

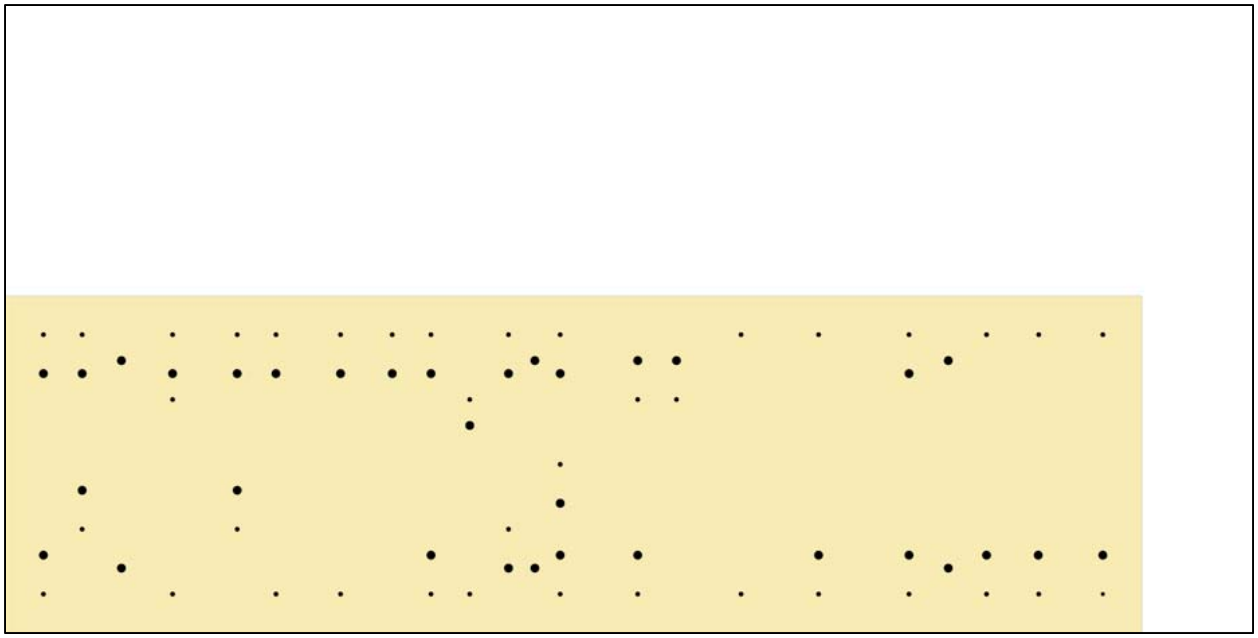
Finally, my little router came with Candle, which works fine for me: https://github.com/Denvi/Candle

I'm sure there's software that's better/easier/prettier/whatever. I chose these because it was convenient for me, and they're free.

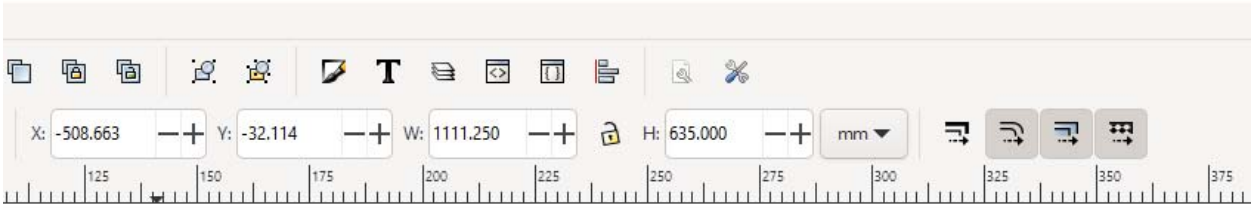
Okay, now that the preamble is out of the way, let's get started! If you don't already have a layout in DIYLC, then open DIYLC and start a new project (If you do have one, skip to the next paragraph). I like to set my workspace to the size of my CNC's workspace (30cm x 18cm, in my case). To do so, go to "Edit, Edit Project Settings" and change your width and height to the appropriate size. From this window you can also set your grid sizing. I like working in inches, for some crazy reason, so I set the grid spacing to .125" (1/8). You can use a board material if you like (I found it helpful), or if your brain can see the layout without having the visual aid of the board, more power to ya. I like to use the standard Blank Board from the Boards folder. Whatever direction you go, you'll want to start your board in the bottom-left corner (trust me). Now, it's time for a tiny bit of math. My turrets were for 1/8" holes, which is 3.175mm. I prefer to use a bit that is bigger than half of the hole, but smaller than the actual size (of course, because if it was bigger, the hole would be too big!), so, I settled on 2mm. Here's the math part: You'll subtract the size of the bit from the size of the hole you want (there's a reasoning behind it, but just trust me). For my example turrets, that means we'll get a 1.175mm. Under the Connectivity folder, you can select the dot. Double-clicking the dot allows you to change the size you calculated. You can now place dots wherever you need a turret. You can also use this time to add any other holes, such as mounting screw holes. Just subtract the size of your bit from the desired size of the hole, and you're good to go! Once you've got your board laid out how you want it, go to "File, Export to PNG." You can now close DIYLC and open Inkscape (and skip the next paragraph).

If you had a previously done layout, delete any items unrelated to the board (potentiometers, wires, switches…anything that isn't a turret/eyelet or the board. Once that's done, select the Dot from the "Connectivity" folder. Place the dot in the center of your turret. You'll subtract the diameter of your CNC's bit from the size of the hole you want (for example, my turrets needed a 3.175mm hole, and my bit was 2mm, so my dot needed to be 1.175mm). Once you have the appropriate sized dot, you can delete your turret symbol. You can copy the original dot you used and paste it until you've got every turret. You can also take the time to mark any holes for mounting screws. I ended up using 2.17 to get 4.17 holes (I'm using #8 screws, and left a little room for play, in case they weren't perfect). Once it's all done, go to "Edit/Select All". Move your board so that it starts in the bottom-left side of the project area. Got to "Edit, Edit Project Settings" and make note of the Width and Height of the workspace. Now, go to "File, Export to PNG." Once you create that file, you can close DIYLC and open Inkscape.

On your last step, you should have created a .png file that somewhat resembles this:

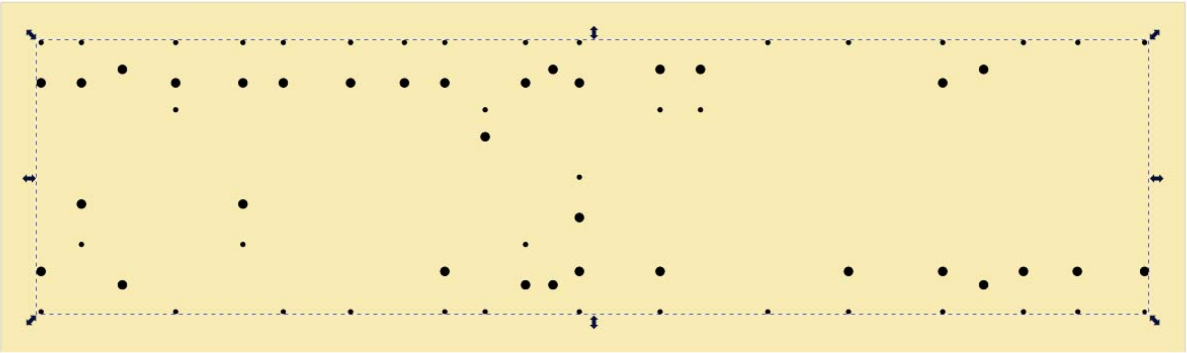


If you've opened Inkscape, you can click the little "New Document" button. From there, I like to set my workspace to the size of my CNC's workspace (sensing a pattern?). Under "File/Document Properties" you can change the size (again, mine is 30cm x 18cm). Next, go to "File/Import" and find the .png file you created in the previous step. Once it's imported click on the image. The top of your screen should look something like this:

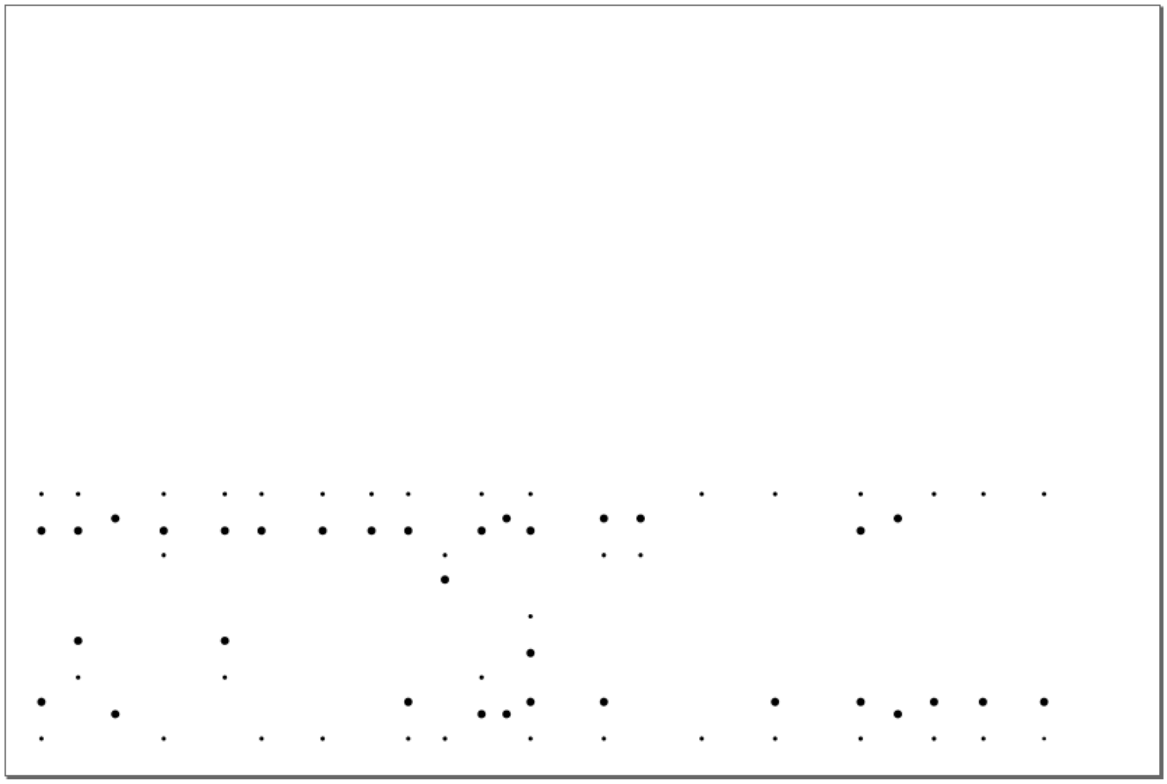


Because we made sure to start in the bottom-left corner, this part is easy: In the X and Y boxes, change those values to "0". Use the dimensions of your DIYLC project from step one to fill out the W and H boxes. If you do that, and your project isn't in the bottom-left, one of two things likely happened: Either you made a mistake, or Inkscape has 0,0 in the top-left instead of bottom-left. To change that, go to "Edit, Preferences" and make sure the box that says, "Origin at upper left with y-axis pointing down (requires restart)" is unchecked. As the text implies, you'll need to restart the program if you change that field, so make sure to save your project before doing so!

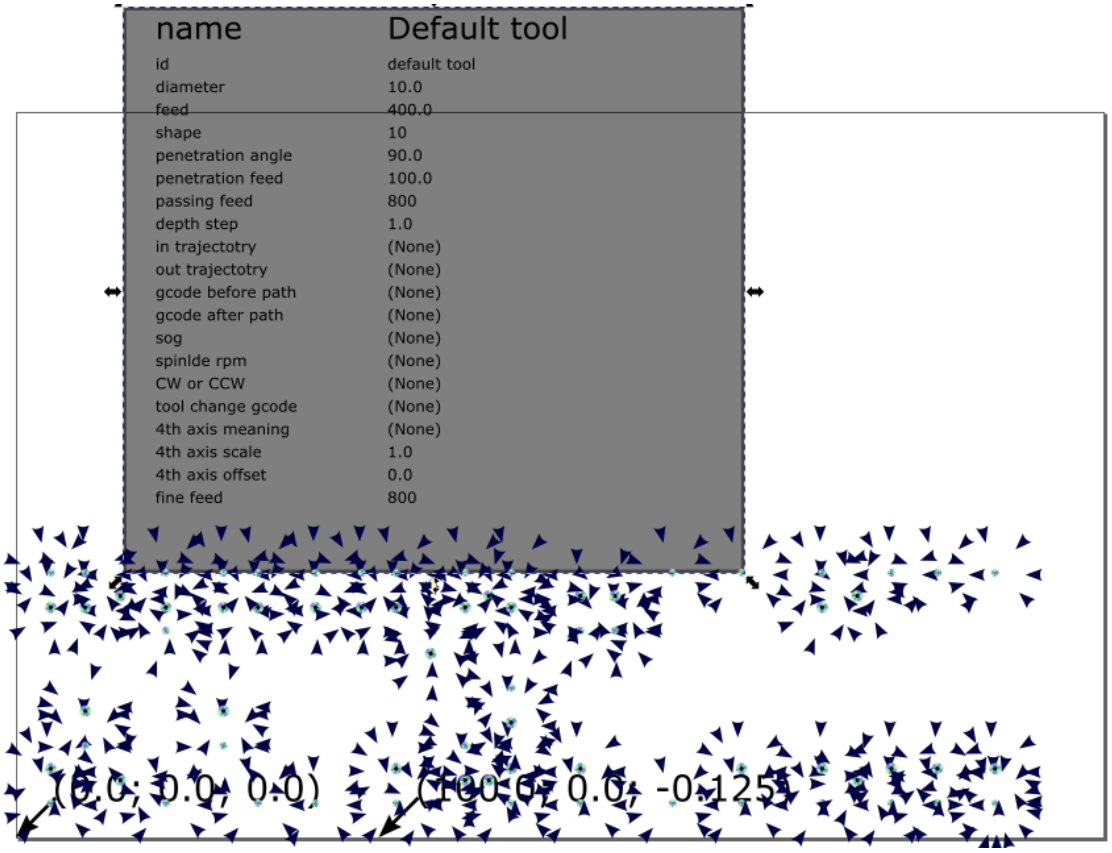
Next, click on the image of your board, then go to "Path, Trace Bitmap." The top right of that box is a preview screen. Without changing settings, you should be able to click the "Update" button and see your array of dots show up in the preview. If that worked, click the apply button and close the "Trace Bitmap" window. You should now see a bounding box around your dots, like so:



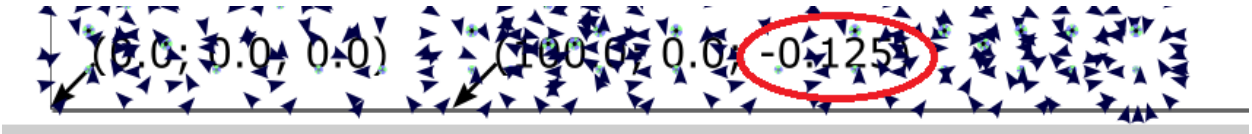
This is good! Now, click outside that box, but still on your image so that it highlights the entire thing again. Click Delete. You should now only have the dots on your workspace, as such:



You might notice that I have several bigger holes aside from the turrets. This is because I like to loop my wires through the board. It reduces strain on the solder joint, plus I think it looks cool. Anyhow, click back in the box to highlight your dots again. There are plenty of steps you can do at this point, but for simplicity, let's do it the easy way. Click "Extensions, Gcodetools, Path to Gcode." On the "Preferences" tab, choose what you want to call the file (while leaving the .ngc at the end), as well as the Directory location where the file will be. Also, there's a field for "Z safe height for G00 move over blank". I believe the default is 10mm, but I change it to 2. It saves a lot of run time. Everything else should be able to stay the same. Click the "Path to Gcode" tab, then click "Apply". You'll get errors about orientation points and a cutting tool, but you can ignore those. There should be a large box with a lot of text. Feel free to shrink that so it doesn't take up your entire screen. Maybe something like this:

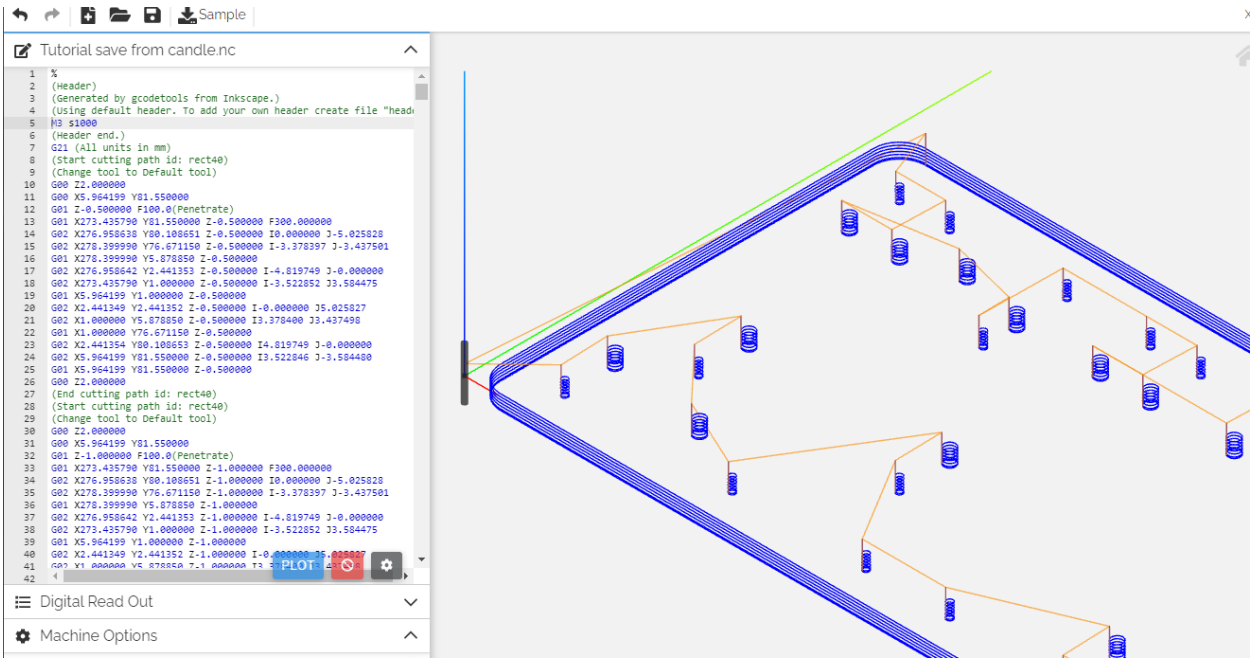


Now is the time to "fine tune" some of the settings. Click the text tool at the left of the screen (illustrated on the left of this page). I don't know if it's necessary, but I like to change the Diameter to the actual tool size (2mm in my case). Because G10 is pretty hard, I also slow the feed to 300. I didn't seem to have trouble. I did bump the passing speed up to 1500 to move things a bit quicker. I also changed depth step to .5. This is how deep the bit will go each pass. I didn't get any burning, nor did my motor seem to bog down. I did experiment with leaving it at 1, and the motor didn't seem happy. The last change I made is at the orientation points. The specific number we want is here:



This is the thickness of the material. Mine was 1/8" nominally (which is 3.175mm). However, after breaking out the calipers which said it was closer to 3.5mm, I ended up settling on 3.7 to get all the way through (and remember to keep this a negative number; you're cutting below 0Z).

Again, click "Extensions, Gcodetools, Path to Gcode." You'll get an error about no paths selected, but ignore it. You should now have a functional G-code! Congratulations! Just to check, I like to go to ncviewer to verify my file. Here's what that might look like (ignore the border on this one; I briefly thought about cutting the board out on the router, but I decided not to waste the time and money on the router bits it would destroy):



As long as you get something that appears to look like the board you want to cut, you can now move on. One thing I found: For some reason, there's no spindle speed specified, and you'll need to manually enter it. You can open the G-code in Notepad. After M3, enter s with some value after it (for example the line may look like: M3 S2000). My spindle will do 10,000 RPM, according to the specs, but I had visions of things flying apart, so I took it slow and steady. I may update this if I find more optimal settings, but this should get you by for now. Anyhow, it's now time to open Candle!

For starters, you'll need to connect your router. Mine has a USB port. Go to "Service, Setting" The first option is "Connection". You may need to hit the refresh button, but the port should show up for you to select. If not, refer to your machine's manual about how to get it to interface with your computer. Once done, close that window.

It may be a little late to discuss, but you'll want to be sure to use the proper kind of mill. I use something with an end like these:

I like to get my material attached at this point. I have a sacrificial piece of mdf on top of the aluminum spoil board, then my G10 gets double-sided tape to attach it to that.

Next, you'll need to find 0,0 for your machine. Using the controls in the "Jog" menu towards the bottom right, move your spindle all the way to the bottom left. I should mention, you'll want to progressively lower the "Step" value so that you don't max our your machine. I actually like to rotate the motors by hand once I get close to the end of travel. Click the "Zero XY" button (shown in red below):

Next, I like to use the easy method for setting the Z axis. I eyeball the bit, and when it's looking close, I put a piece of paper down and lower the mill bit by bit until I get a fair amount of resistance when pulling the paper out. I know; very technical, but it works. Once you've found the zero position, click the "Zero Z" button (shown in green to the left).

Now that the machine is calibrated and ready, you can open your file by clicking "File, Open" and selecting the location where you saved your file. Now, all that's left is to click the "Send" button. Unless something goes horribly wrong (and believe me, I had a couple of those moments). At that point, the handy "Abort" button is there.

Assuming all has gone well, congrats on your new CNC board! If it didn't go well, congrats on your learning experience!