

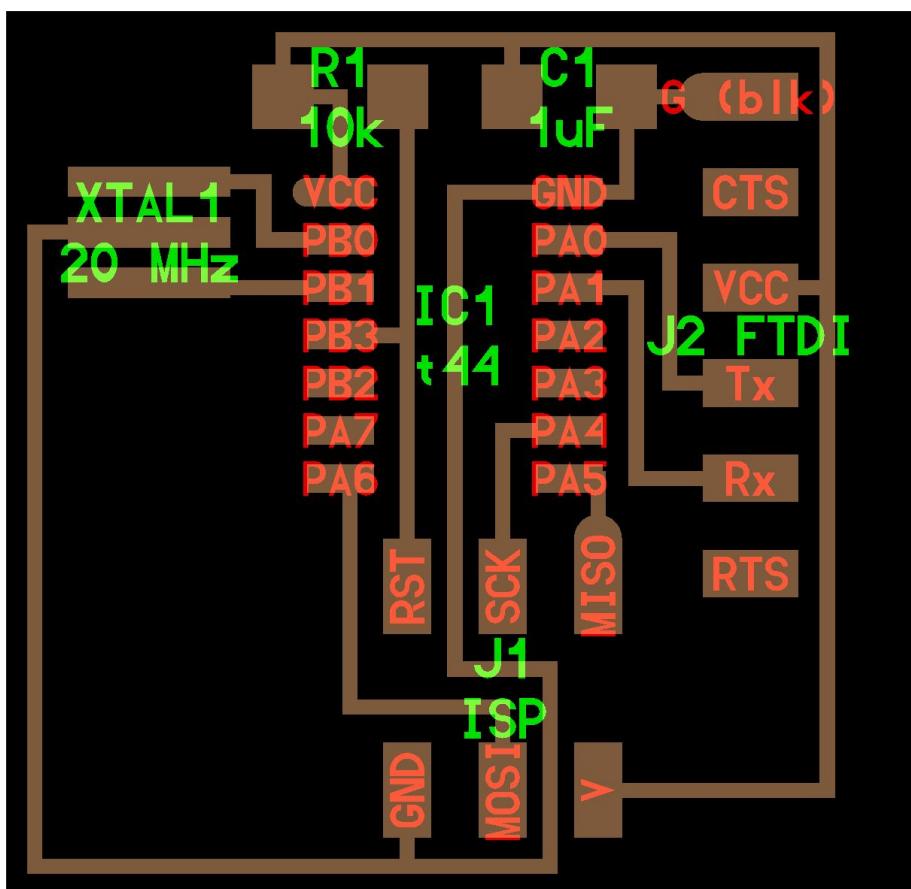
Instructions for designing the HelloWorld circuit board using Autodesk Eagle 8.6.0

FABLAB BRIGHTON 2018

These instructions take you through step-by-step the process of creating the full circuit board design using Eagle 8.6.0. If you're using a different version of Eagle, the icons might be slightly different, but using commands will still work. A summary of main steps is:

1. Import components library
2. Add all components into schematic view
3. Wiring up components in schematic
4. Switch to board view
5. Move components to create the basic board layout
6. Create routes between components to join them all up!

But first, here's Neil's original HelloWorld board as a reference



Components needed

Here is a list of the components needed, the specific component and then how to bring the FAB library into Eagle. Understanding these components and specifically which ones we will be using is important.

Here's a summary of how to find the correct components. Make sure you select the FAB library on each occasion before searching. Then you can simple search for these specific terms and you'll find the components directly.

Component	Search for
IC1t44	SOIC14
J1 ISP	PINHD-2X3-SMD
J2 FTDI	FTDI-SMD-HEADER
Resistors	R1206
Capacitor	C1206FAB
Resonator	EFOBM

Resistor

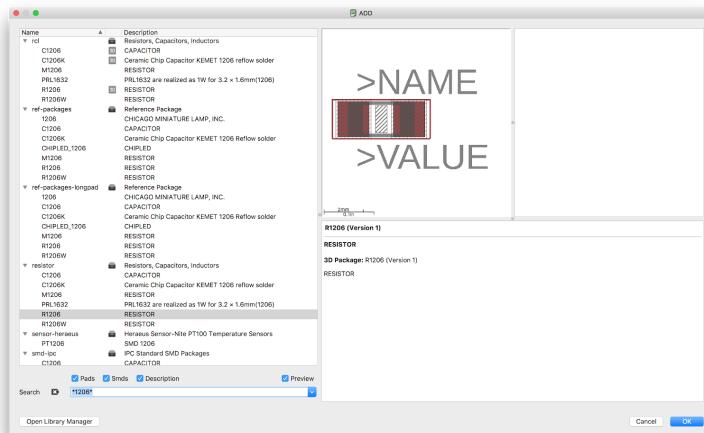
Name on schematic: R1

Value: 10k

Package size: 1206

Eagle library: Resistors, Capacitors, Inductors

Eagle name: R1206



Capacitor

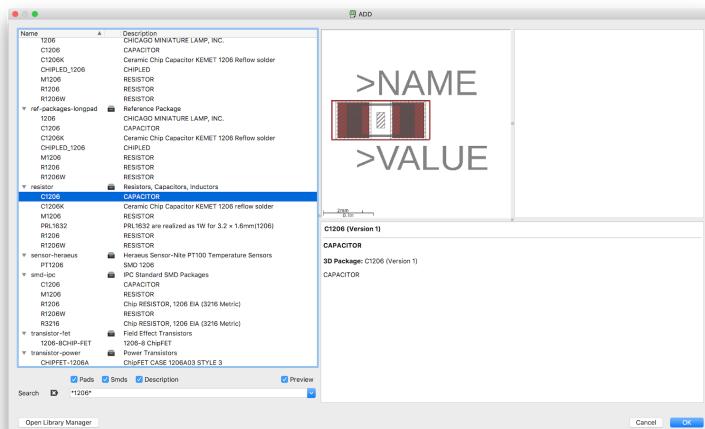
Name on schematic: C1

Value: 1uF

Package size: 1206

Eagle library: Resistors, Capacitors, Inductors

Eagle name: C1206



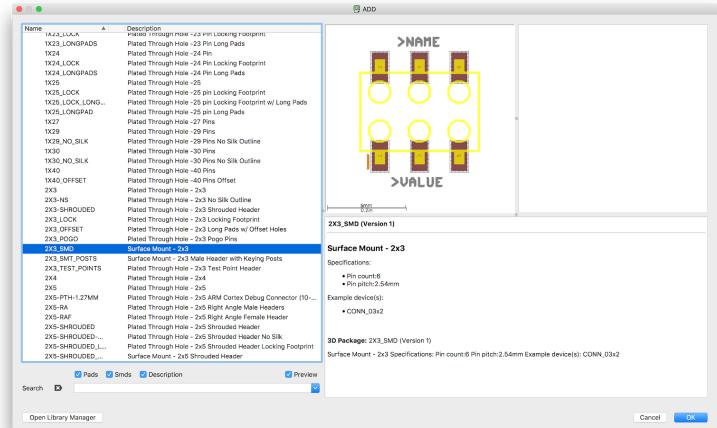
ISP Header

Name on schematic: J1 ISP

3x2 header pins

Eagle library: Sparkfun Connectors

Eagle name: 2X3_SMD (Version 1)



FTDI Header

Name on schematic: J2 FTDI

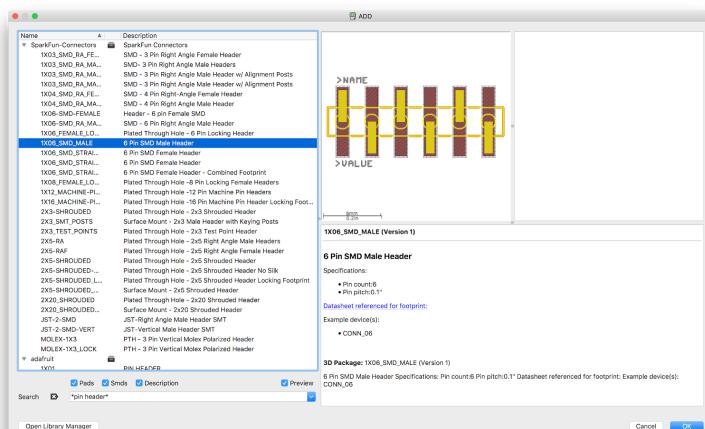
Value: 6x1 pins

Package type: 6 Pin SMD Male Header

Eagle library: Sparkfun Connectors

Eagle name: 1X06_SMD_MALE (Version 1)

Note: Our headers have pins that alternate on each side



Pins should be labelled:

- 1 GND
- 2 CTS
- 3 VCC
- 4 Tx
- 5 Rx
- 6 RTS

Crystal

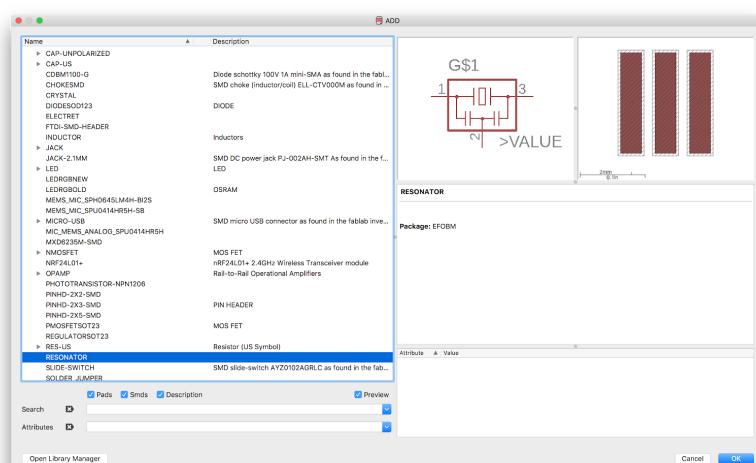
**Name on schematic: XTAL1
20MHz**

Value: 20 MHz

Package type: ?

Eagle library: Fab

Eagle name: RESONATOR



Chip

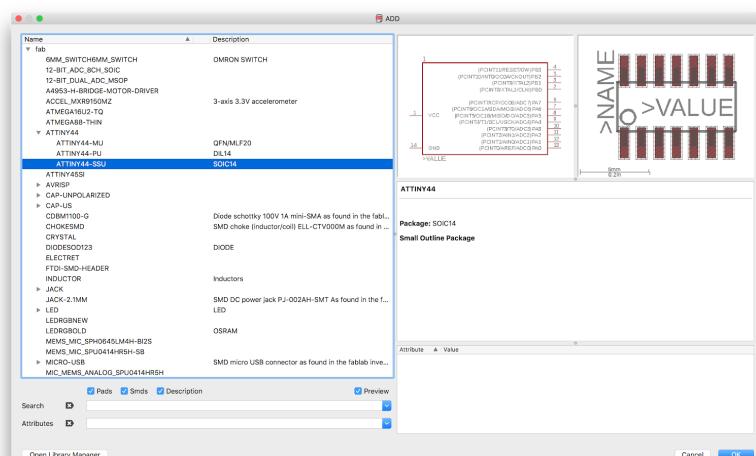
Name on schematic: IC1 t44

Value: ATTiny 44

Package type: SOIC14, Small Outline Package

Eagle library: Fab

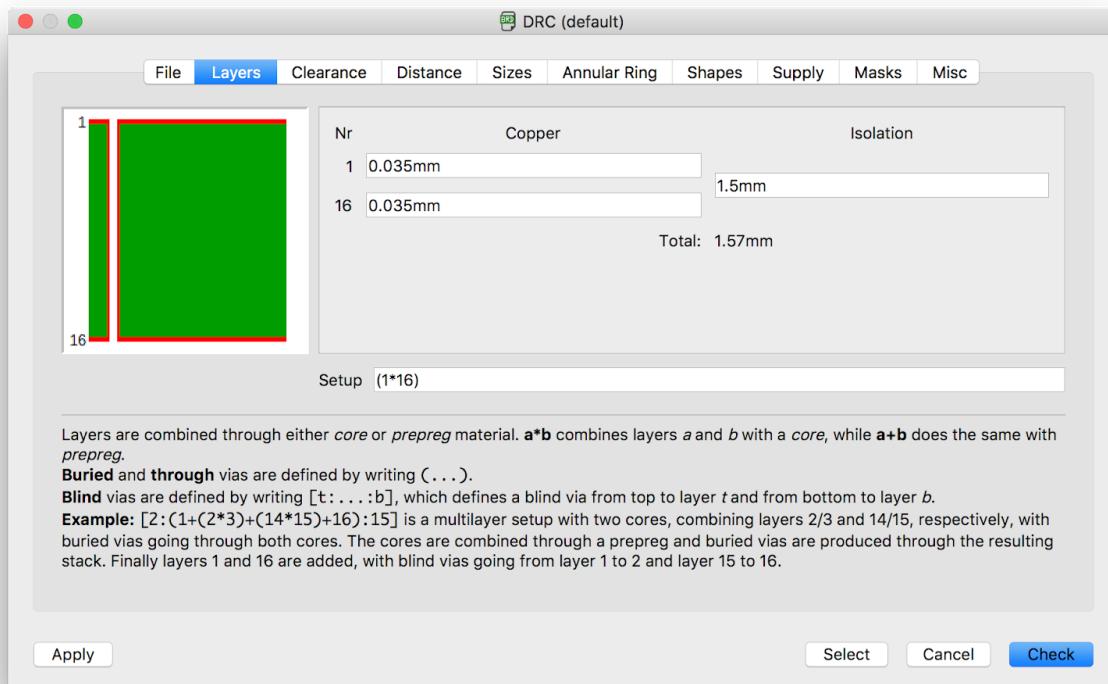
Eagle name: ATTINY44-SSU



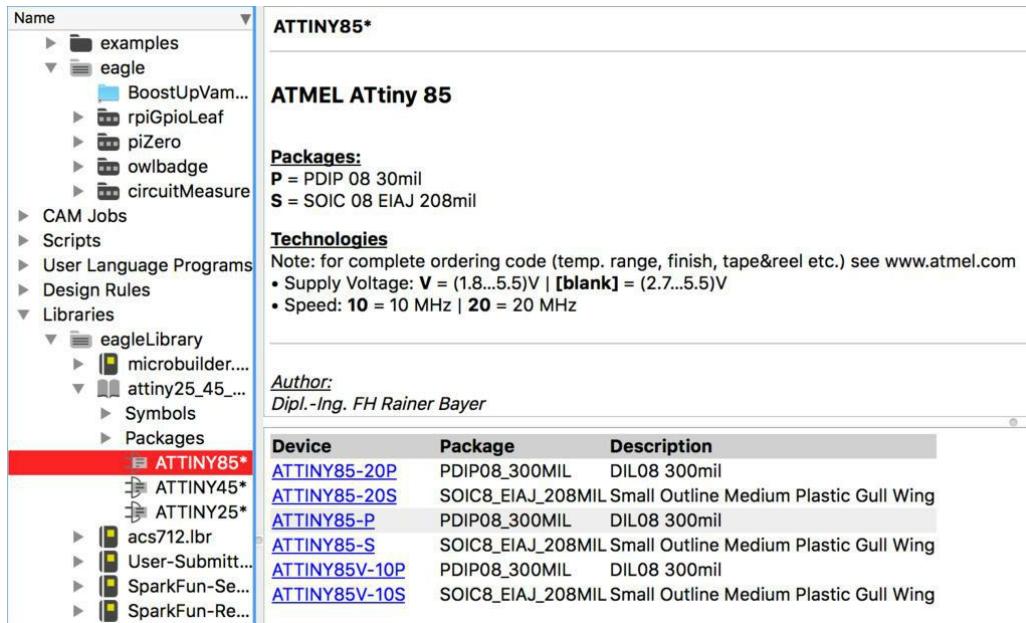
Design rules for Board layout in Eagle

What do we need to set up for producing our boards on our MDX-50?

On the Board view in Eagle, open the Design rules window (Tools menu > DRC...)



-ADDING A LIBRARY TO EAGLE CAD-

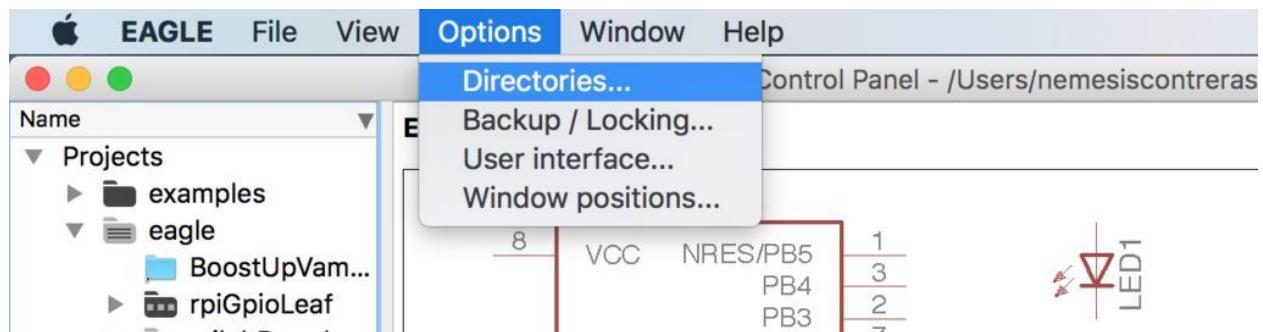


Step 1: Download the Library You Want to Use

Step 2: Create a Folder to Place the Library in

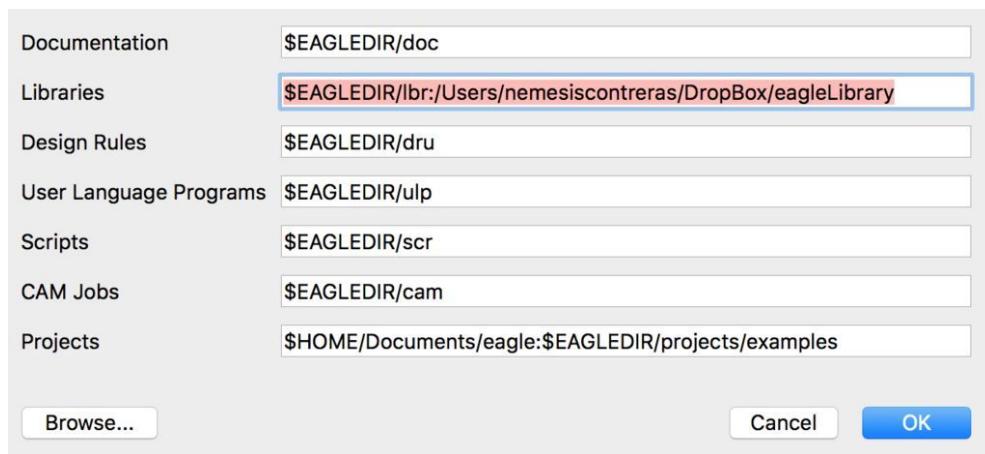
Step 3: Drag the Library to Your Folder

Step 4: Tell Eagle CAD Where to Look for the Library

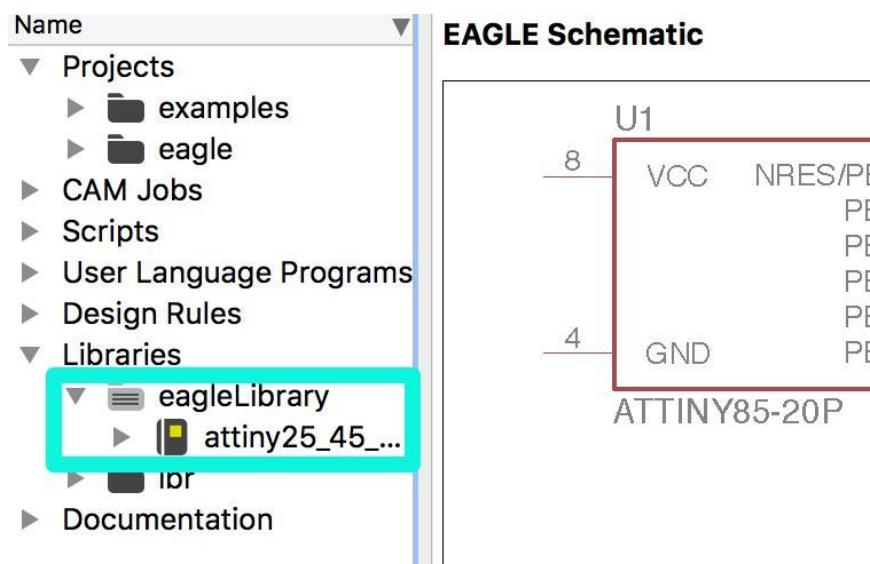


Open Eagle under Options->Directories

Step 5: Under the Directory Type the Library Location

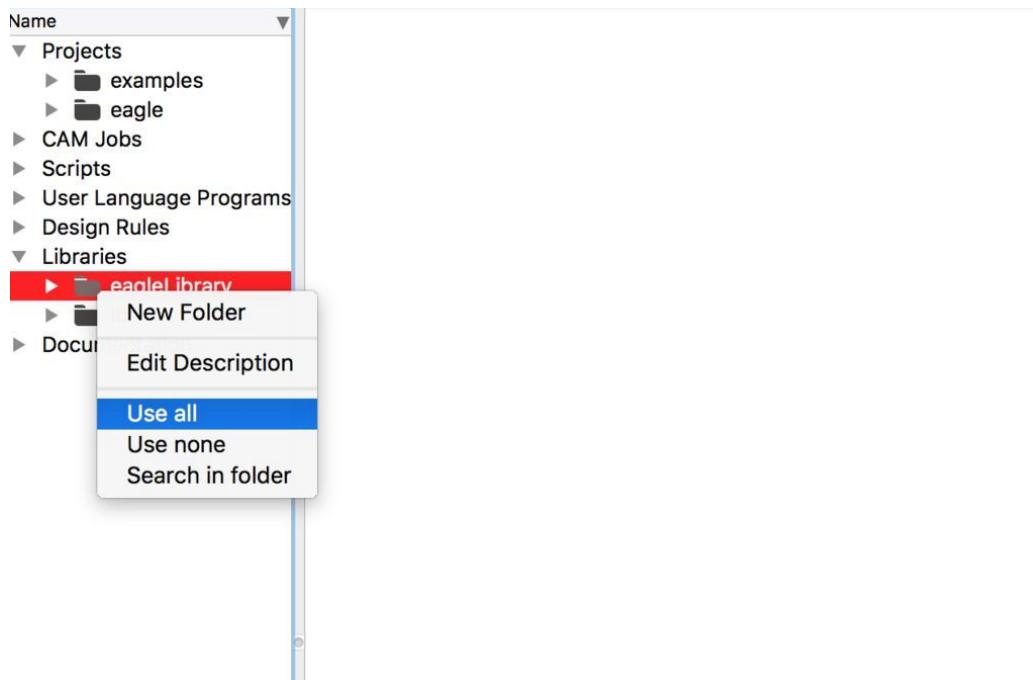


Step 6: Check



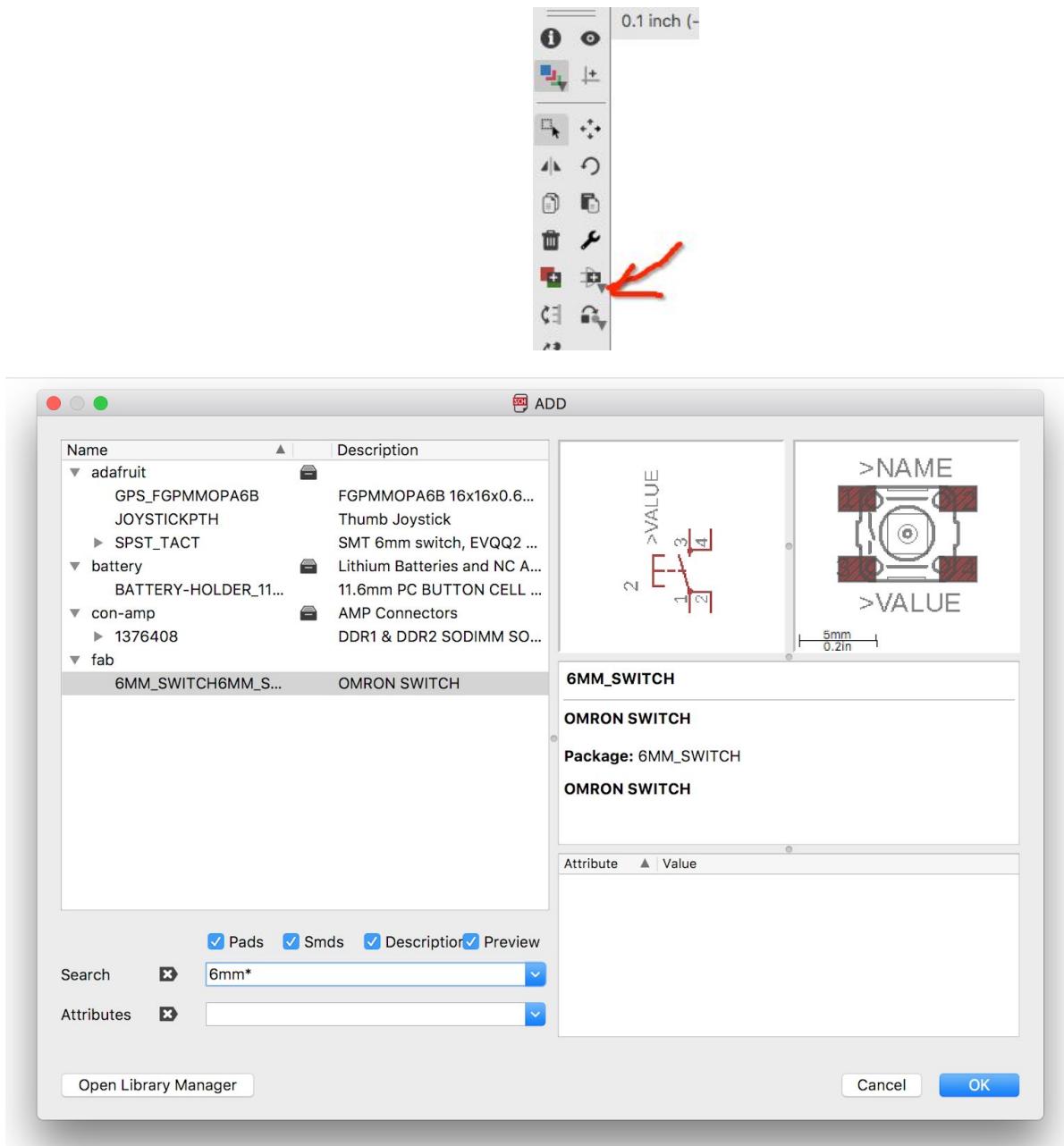
If done correctly you should now see the folder with the library show up

Step 7: Include it



Right click your library the select Use All so Eagle CAD knows to use the components in this library

-SEARCHING COMPONENTS IN THE “ADD” COMMAND-



There are some ways to search for the same component:

Type its plain name and pray for it to be found Ex: “Switch” ;

Add a * to the end of the work (As old DOS, “*” meaning “All”) Ex: “Switch*”;

Add a * to the beginning of a word. Ex: “*Switch”;

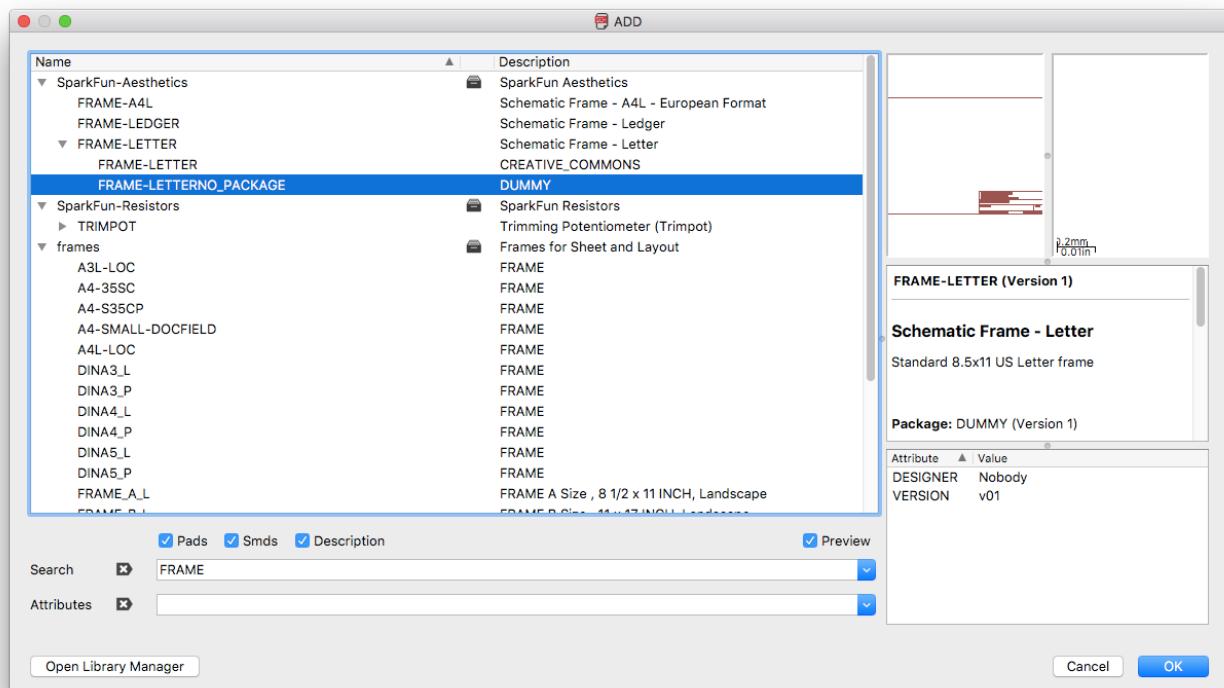
Add a * to the beginning and another to the end of a word. Ex: “*Switch*”;

You can also search for a characteristic of the part. Knowing the Switch I wanted was a 6mm one, searching for “6mm*” found it.

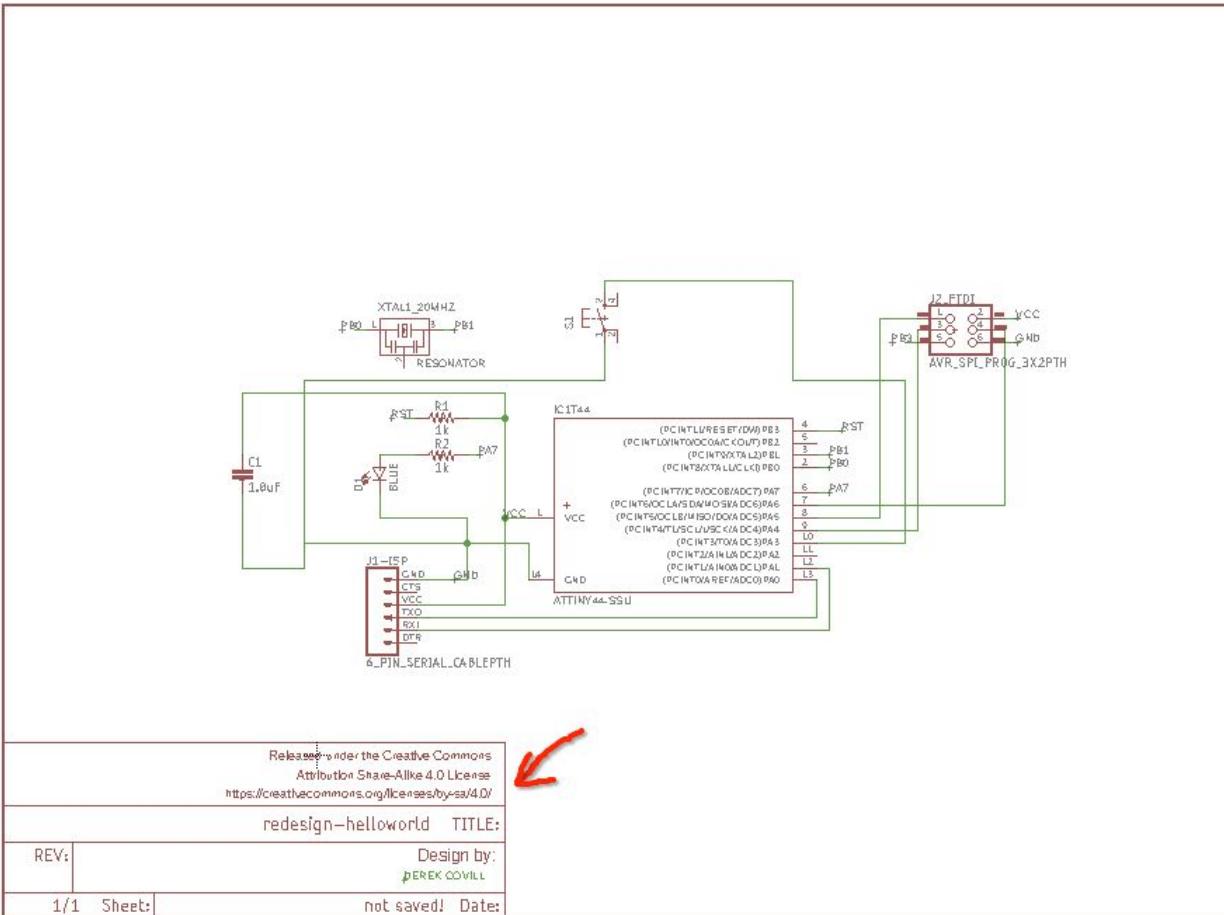
Adding a Frame

You can add a frame (otherwise known as a title block) to your schematic. This will help to identify it as yours, with date, revision details etc. It's good practice.

It's added like a part (in the Sparkfun-Aesthetics library) so search for “FRAME-LETTER”.



Once you've found it, move the frame around so it is positioned around your parts.

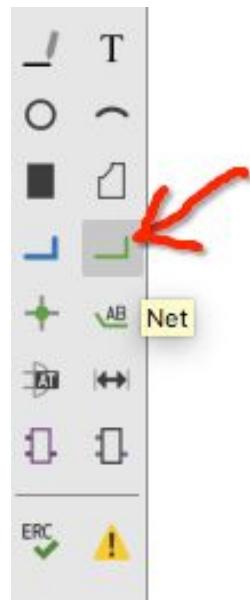


Then add your name with the text tool.

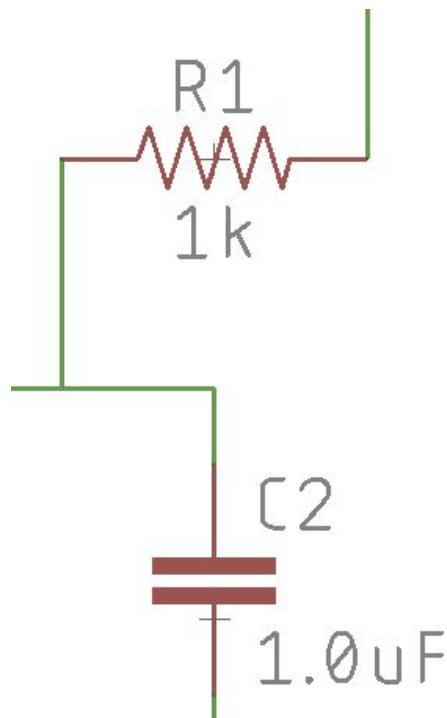


Wiring up the schematic

Once you've added in all your components, you need to connect them using the NET tool as shown below.

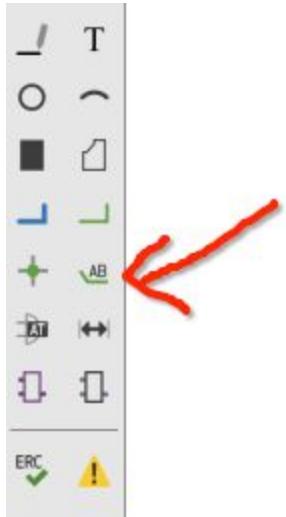


To do this, click the connection part of the first component you want to connect, then using a series of clicks to define anchor points, find your way to the connection point of the other component. Here's an example of a path connecting R1 to C2.



Adding labels to your connections

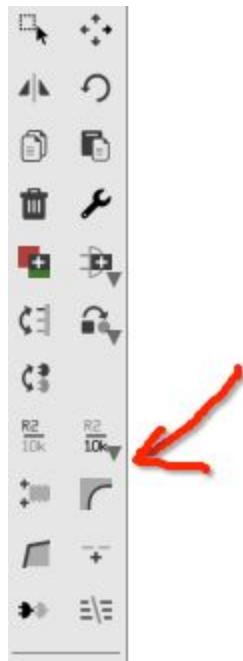
Note that it is good practice to add labels to your components and to the pins using the LABEL tool (it's located below the net tool, with a little AB on it).



Once you've placed the label, you can name/rename it using the NAME tool. Note that if you name two lines/contacts the same (e.g. GND), then they will create a virtual connection between them. This means that you don't actually have to create a NET link between them as outlined above. This can be useful if you have a very busy/complicated board.

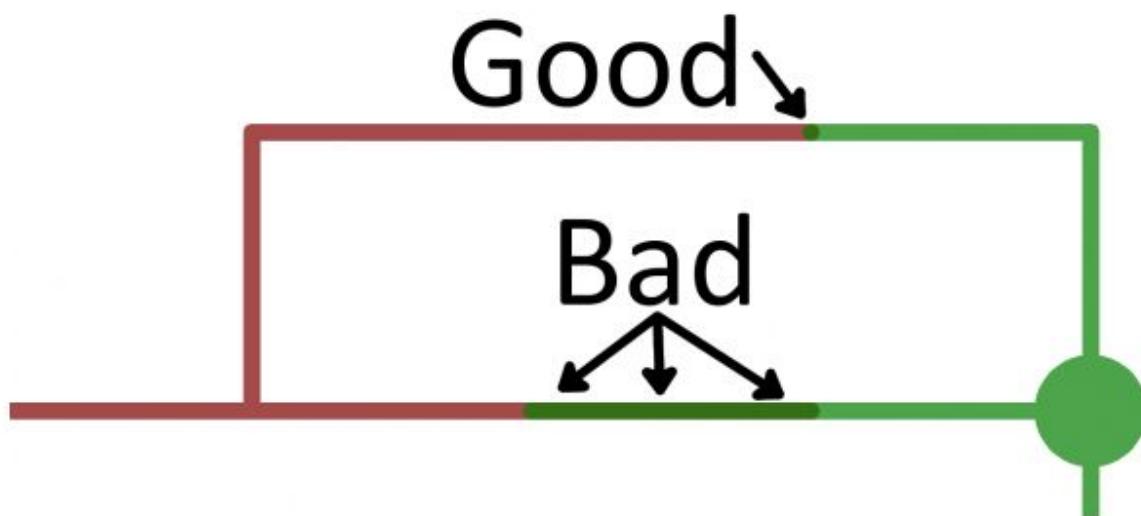


Note also that you can change the value of a component (e.g. the resistance of a resistor) by using the VALUE tool which is next to the NAME tool.

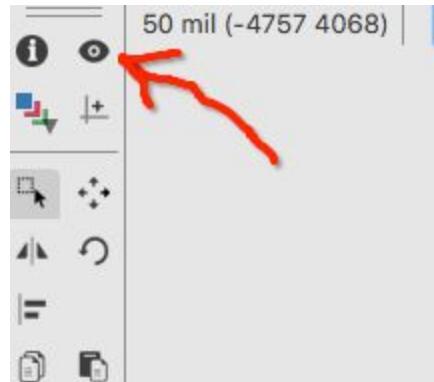


Testing your connections

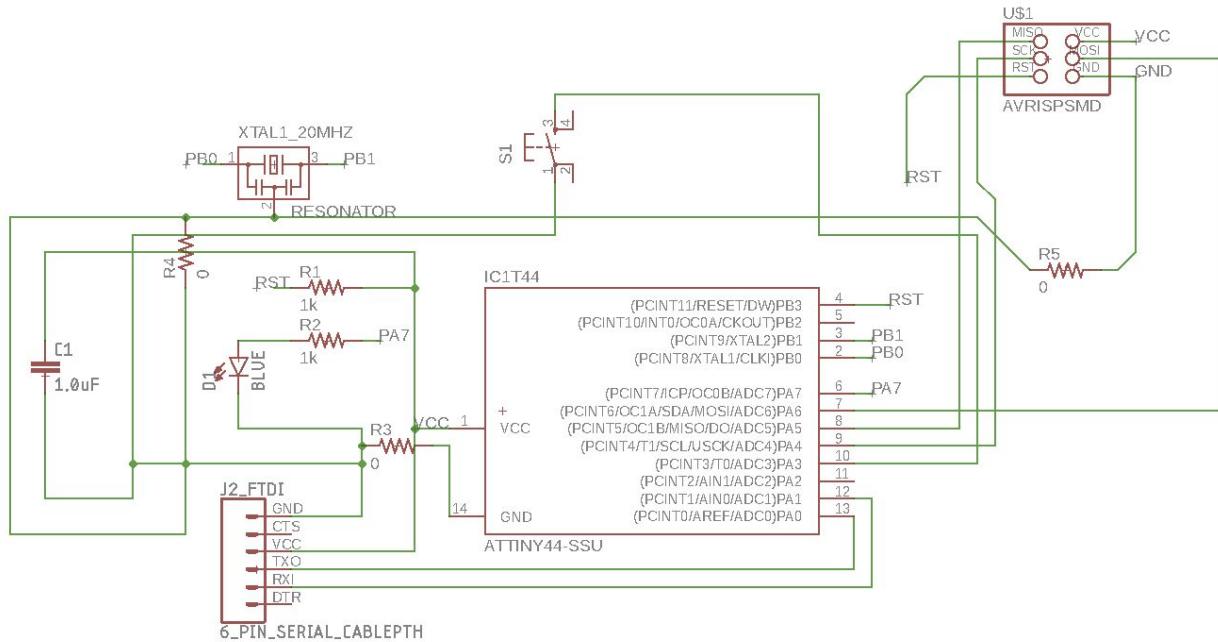
Test your connections by moving them around to see if any of the connections are broken. If so, connect them manually, or delete them, redraw them and then retest them. The thing to really look out for is an overlap between connections like this.



The SHOW tool shown below is useful for verifying that pins across your schematic are connected correctly. If you use SHOW on a net, every pin it's connected to should light up. If you're dubious of the fact that two like-named nets are connected, give the SHOW tool a try. SHOW-ing a net connected to GND, for example, should result in a lot of GND nets lighting up.

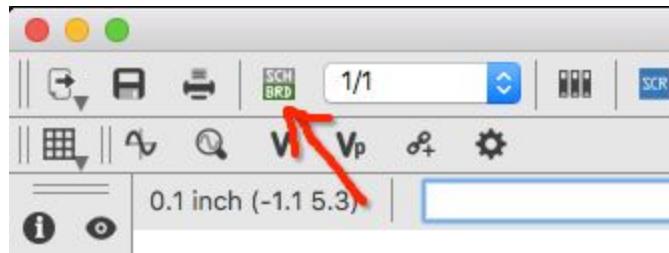


Once all your components are connected, your circuit should look something like this.



Moving to BOARD view

To switch from SCHEMATIC to BOARD view (and back again if you want to check forward and backwards), click on the GENERATE/SWITCH TO BOARD shortcut then All of the parts you added from the schematic should be there, stacked on top of each other, ready to be placed and routed.



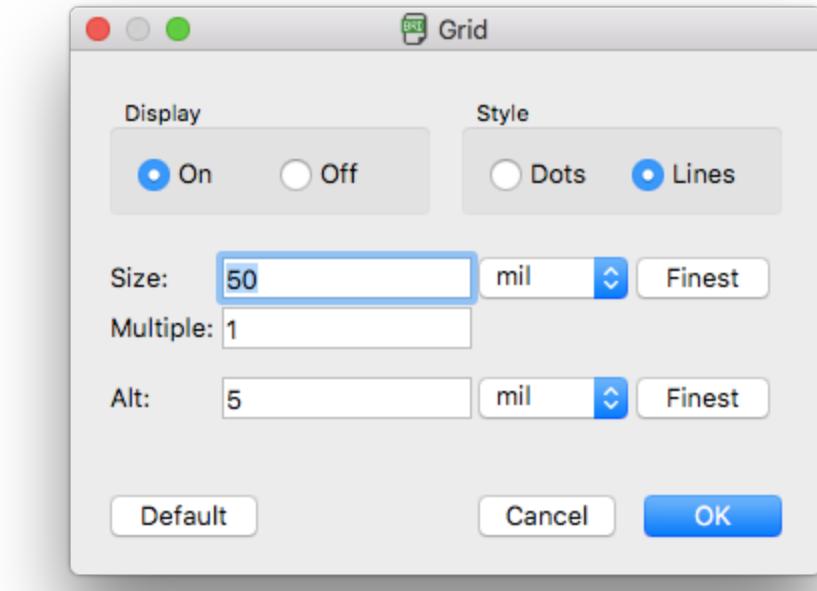
Selecting From Overlapping Objects

Some useful tips from Sparkfun. Since the board view is entirely two-dimensional, and different layers are bound to overlap, sometimes you have to do some finagling to select an object when there are others on top of it.

Normally, you use the mouse's left-click to select an object (whether it's a trace, via, part, etc.), but when there are two parts overlapping exactly where you're clicking, EAGLE doesn't know which one you want to pick up. In cases like that, EAGLE will pick one of the two overlapping objects, and ask if that's the one you want. If it is, you have to left-click again to confirm. If you were trying to grab one of the other overlapping objects, right-click to cycle to the next part. EAGLE's status box, in the very bottom-left of the window, provides some helpful information when you're trying to select a part.

Using the grid

The grid should be visible in the board editor. You can adjust the granularity of the grid, by typing GRID into the command line. Here is a recommended grid setting. EAGLE forces your parts, traces, and other objects to "snap" to the grid defined in the *Size* box. If you need finer control, hold down ALT on your keyboard to access the **alternate grid**, which is defined in the *Alt* box. Here are the default values which work quite well, but if you need really fine control, then set the size to 10 mil instead.



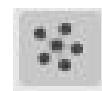
Moving parts

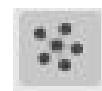


Using the **MOVE** tool  you can start to move parts within the dimension box. While you're moving parts, you can **rotate** them by either right-clicking or changing the angle in the drop-down box near the top. Note also that when dragging, you need to select the CROSSHAIR for the part.

The way you arrange your parts has a huge impact on how easy or hard the next step will be. As you're moving, rotating, and placing parts, there are some factors you should take into consideration:

- **Don't overlap parts:** All of your components need some space to breathe. The green via holes need a good amount of clearance between them too. Remember those green rings are exposed copper on both sides of the board, if copper overlaps, **streams will cross** and short circuits will happen.
- **Minimize intersecting airwires:** While you move parts, notice how the airwires move with them. Limiting criss-crossing airwires as much as you can will make routing *much* easier in



the long run. While you're relocating parts, hit the RATSNEST button  or just type RATSNEST to get the airwires to recalculate.

- **Part placement requirements:** Some parts may require special consideration during placement, e.g. orientation.

Using the Route Tool



To draw all of our copper traces, we'll use the ROUTE tool , and a typical width of 16mil is better than 10mil.

Then start your route by left-clicking on a pin where an airwire terminates. The airwire, and connected pins will "glow", and a red or blue line will start on the pin. You finish the trace by left-clicking again on top of the other pin the airwire connects to. Between the pins, you can left-click as much as you need to "glue" a trace down.

Move around your board connecting the pads together, but also noting that it might be best to rotate your parts or move them around to MINIMISE THE NUMBER OF CROSSINGS.



The RIPUP tool (next to ROUTE TOOL) is SOOOOO useful for deleting parts of your tracks if you want to redo them or tidy them up. Simply type **RIPUP** in the command line.

It's good to use the **RATSNEST** tool often to reset the original links so they're a bit more tidy and you to see the connections more clearly.

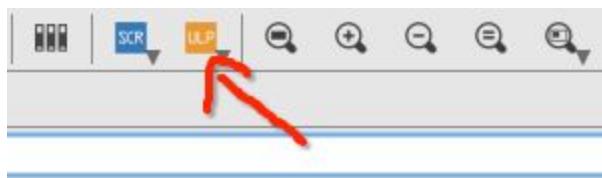
It is likely that you'll occasionally need to use a **0 Ohm resistor** to jump across the tracks. To do this, in SCHEMATIC VIEW you can select an existing resistor, then copy, paste and move it to a clear space. You'll need to figure out what traces you're jumping, so that you can link it into the correct trace location. You can then change the resistance value by right clicking on the resistor and selecting PROPERTIES, then changing the value to 0.

Note also that sometimes there are issues with the connections (i.e they look connected, but two adjoining traces don't actually join). In these cases you can try to repair them, or simply delete the existing trace and redraw them.

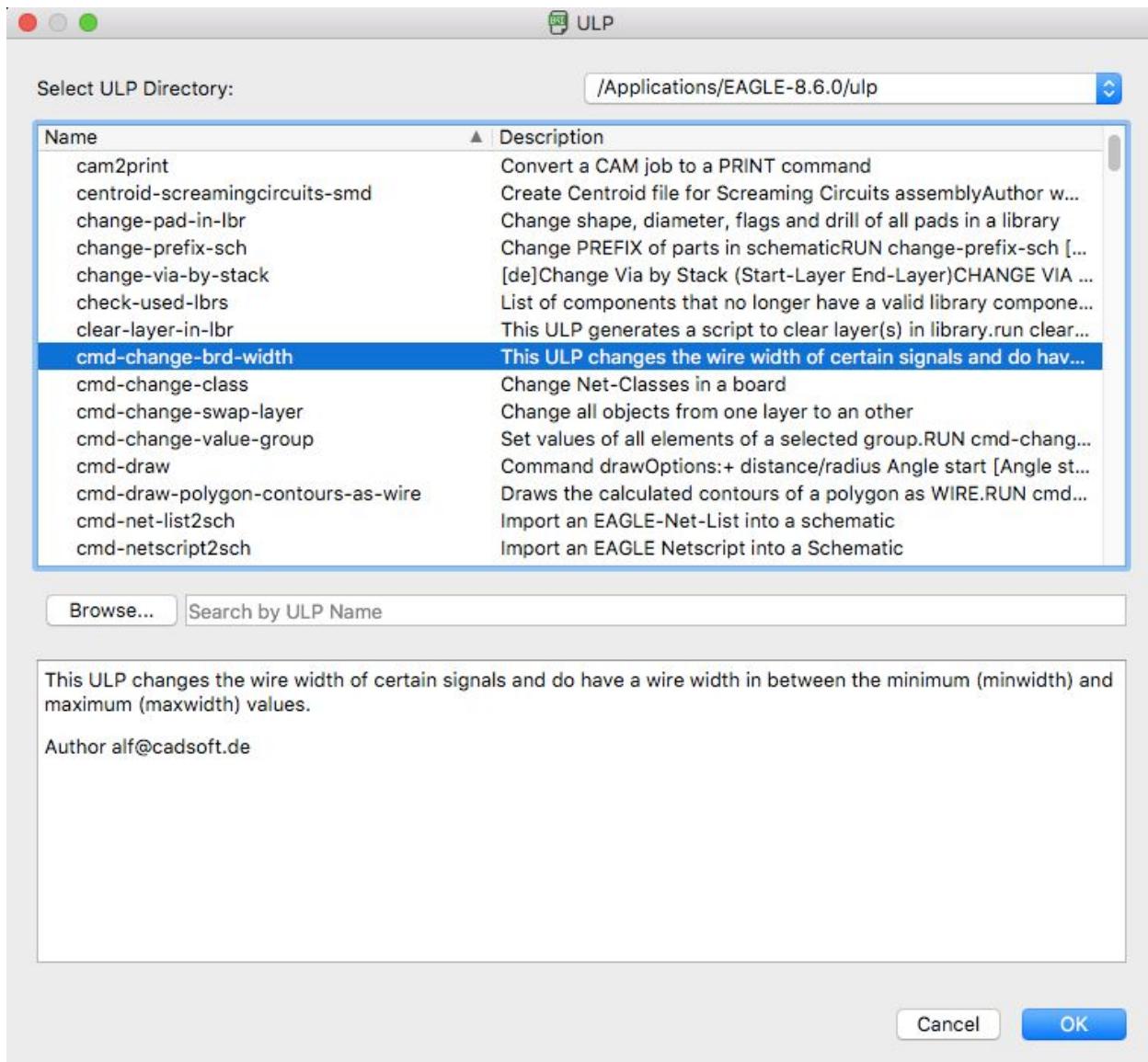
Finally, once you do a **RATSNEST** and there are no unconnected traces you're good to go. Note that if you've used a 0 Ohm resistor, ratsnest will show up (in the information bar at the bottom) that there are airwires still left to be joined. I am assuming that these can be ignored, since it hasn't registered that the 0 Ohm resistor will jump the tracks and keep the current flowing unaided!

If you want to change the trace width (and other parameters), you can do this retrospectively by typing **CHANGE** into the command line, then **WIDTH**, then selecting the trace width...then you simply go round and select each trace and they will update. The default is 10, but **16** is probably best.

If you want to change the width of ALL traces, then select everything on the board (or you can use **CMD+A**, or **CTRL+A**). Click on the run ULP icon (there's no command for this that I can find).

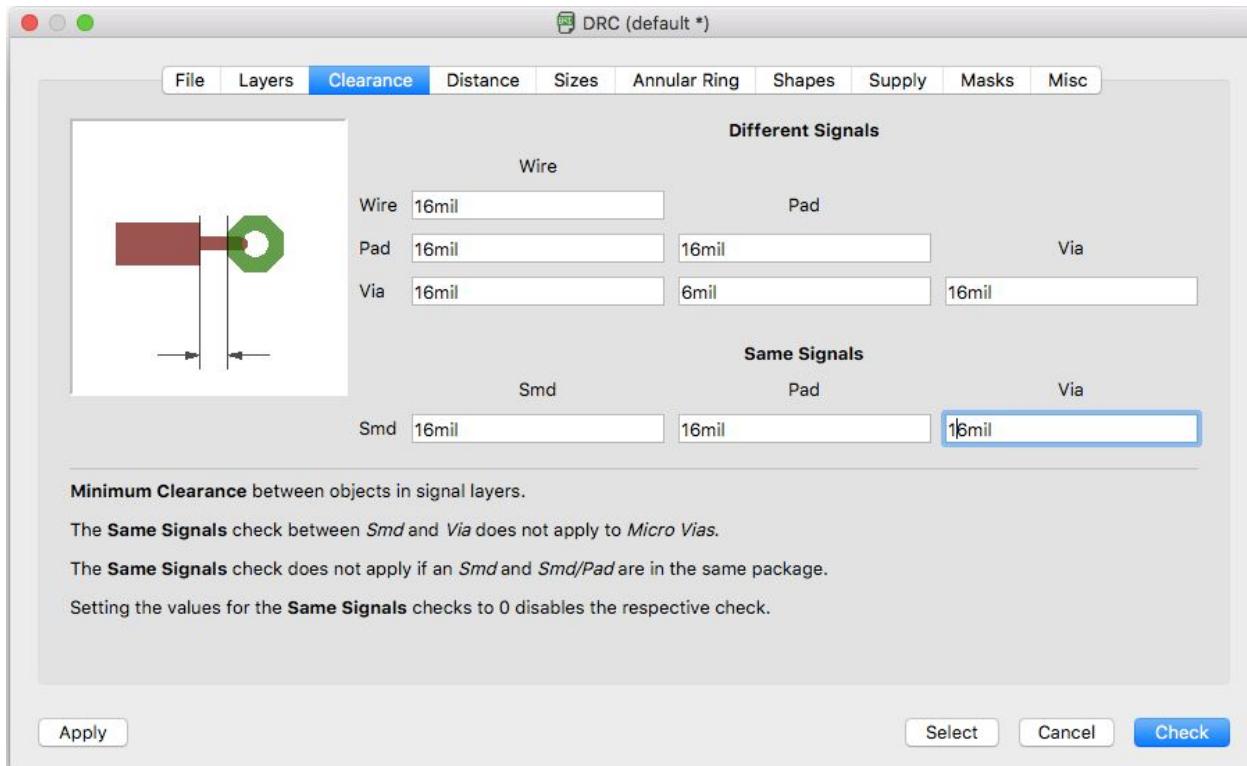


Then select **cmd-change-brd-width**.



Then select the **ADD all-->**, and entire a new wire width (e.g. 16mil) and click OK.

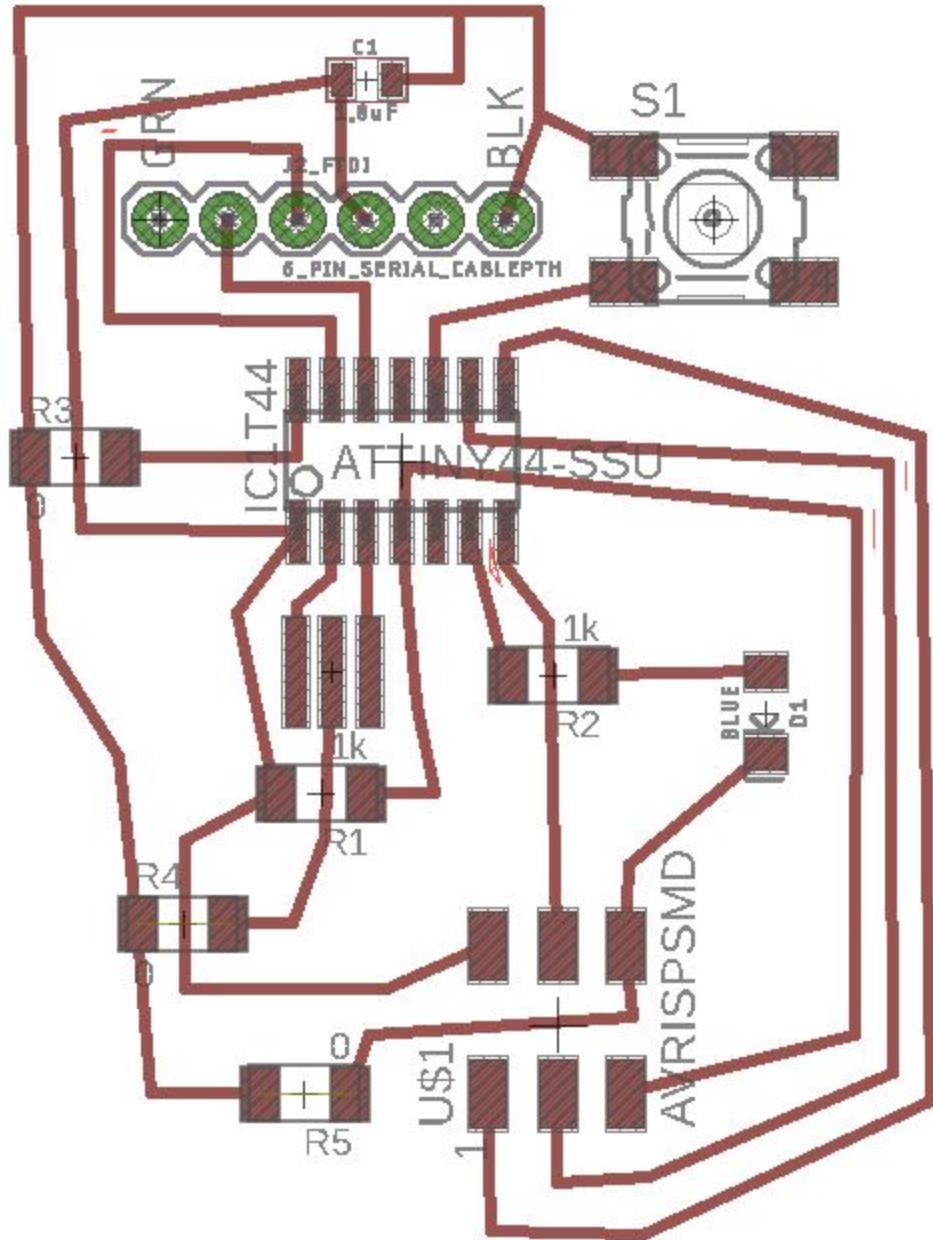
You can then do a thorough design rule check, by typing **DRC**, going to the **CLEARANCE** tab at the top, and entering all your design rules (typically 16mil is good all round here). Note that in the **FILE** tab you can also load a preset set of design rules.



Adding Copper Pours

Copper pours are usually a great addition to a board. They look professional and they actually have a good reason for existing. Not to mention they make routing *much* easier. Usually, when you're adding a copper pour it's for the ground signal. So let's add some ground pours to the design.

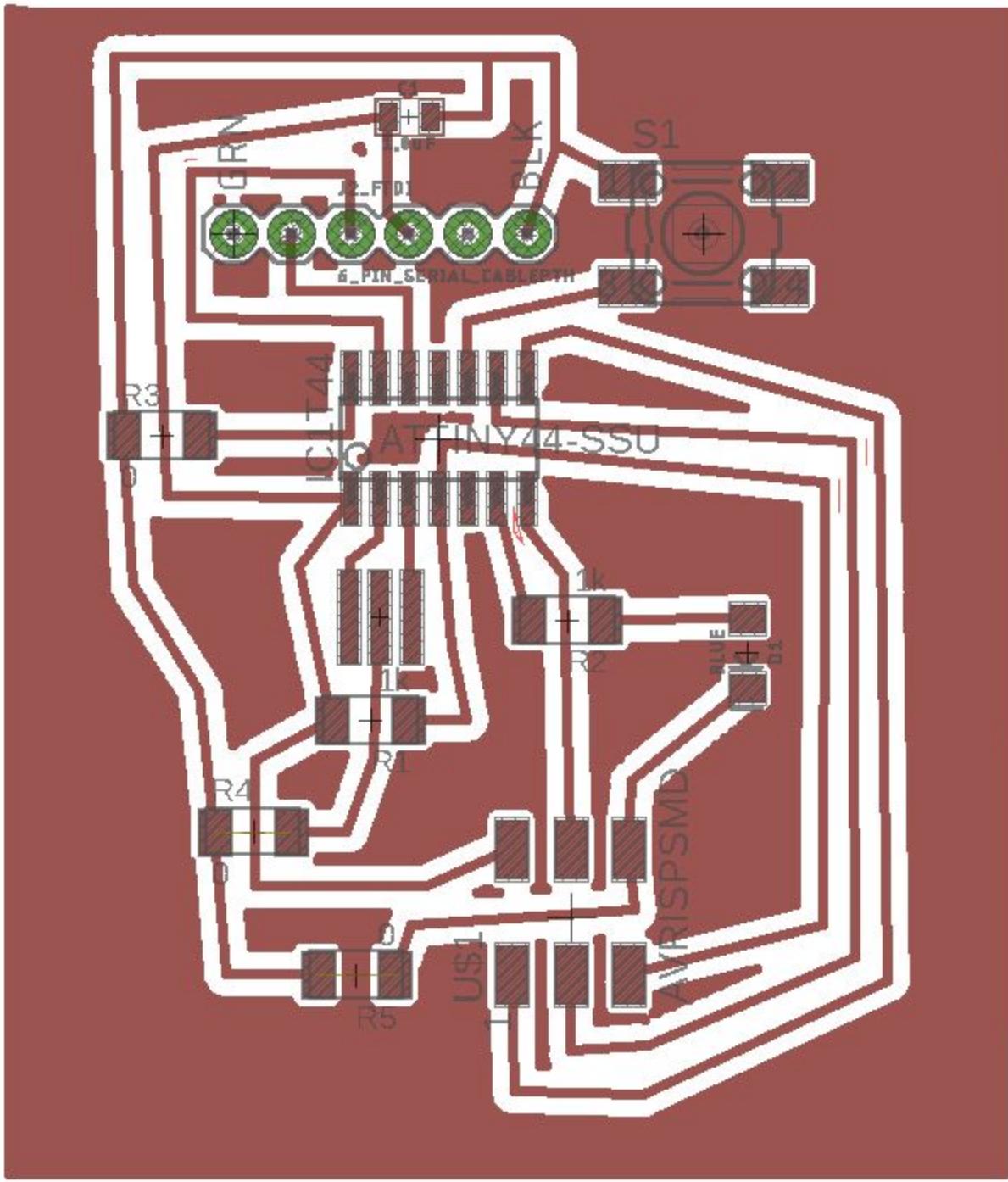
Start by typing POLYGON into the command line. Then simply draw a box around your circuit by clicking 3 corners and double clicking when you get to the final corner. Here's what it will look like.



And we simply want to name the box GND, using the NAME command. Then when we RATSNEST, the border should be filled as shown below. Note that it can be hard to tell what is and isn't connected to the ground pour. If you see a black gap separating a pad and the pour, there is no connection. If you see some traces forming a "target" over the pad, there is a connection from the pour to that pad.

If you ever want to **hide the polygon** (it's hard to see other stuff with it on there), use the RIPUP tool on the polygon border you just drew. Don't worry, the polygon is still there, just hit ratsnest to bring it back.

So here's the final board, ready to be converted to .png format for milling!



Now to export.

We can do this in two ways:

1. Export as .dxf
2. Export as high resolution image (.png) and also save as .pdf.

- a. I suggest you check your board layout first, by actually printing the .pdf on paper, and placing the chip onto it to make sure they match!
- b. Open both files in PS and copy .png into .pdf so they are on separate layers.
- c. Use pdf as template for real size (since this is true scale), then reduce the size of high res .png to bring it down to the pdf size so they overlap each other. This will leave a true scale, high resolution image of your board.