

How to use Eagle to Create a Schematic and Printed Circuit Board

Contents

Get Ready to Work [1](#)

Start Placing Symbols [4](#)

Stealing Symbols from Another Schematic [8](#)

Lay Out the Parts [10](#)

Rule Check the Schematic [16](#)

Laying out the Printed Circuit Board [18](#)

Fix Ground Islands [22](#)

Putting Text on the Board [25](#)

Printing the Printed Circuit Board [27](#)

Printing Double-sided PCB and Silk Screen [27](#)

Creating A Component [27](#)

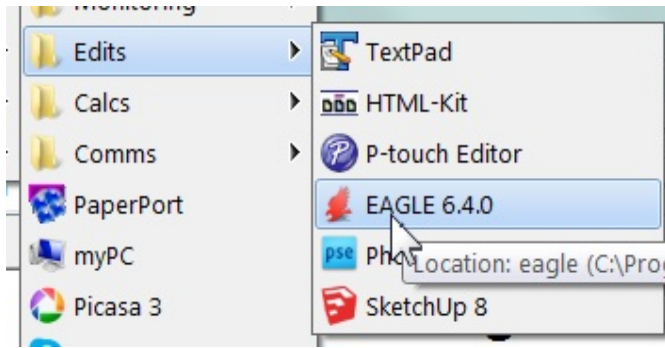
Random Notes [28](#)

Eagle is a little 'different' from other software in it's overall operation which makes it kind of hard for me to remember between long spaced out sessions of activity. This guide will explain how I designed a circuit and PCB to allow doing the same again in the future without having to relearn the whole interface again.

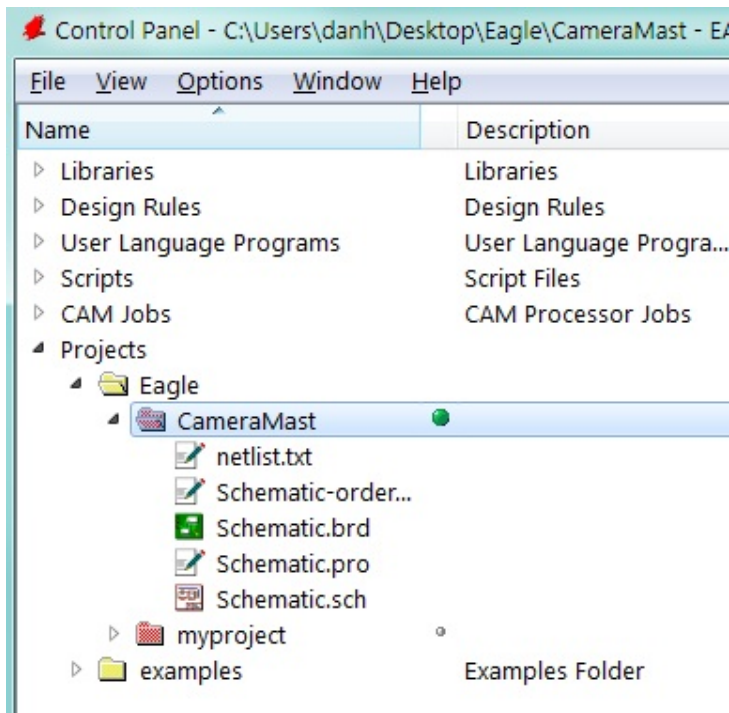
Note: you can type 'assign' in the command line to see what hot keys are mapped to what commands to make navigation easier. For example ctl-M is move, ctl-G is group, and ctl-D is delete.

Get Ready to Work

Start the program:



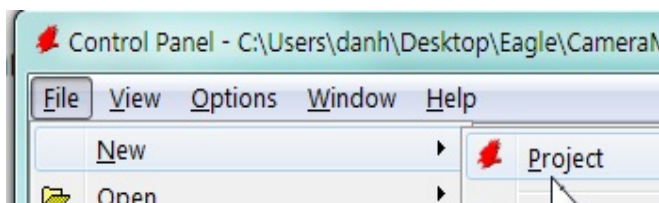
You will see something like:



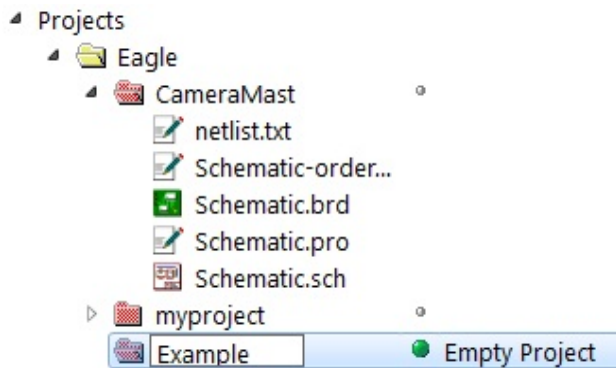
Although I didn't do it for these example, you may want to set the background to black to make it easier to see airwires:

Options | User Interface and select Background: Black (it won't print with a background which is nice).

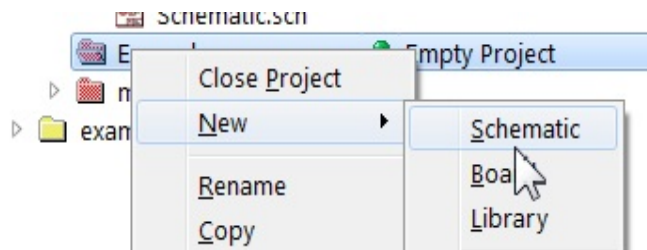
Open an existing project, or create a new one:



and type in the name of the new project:

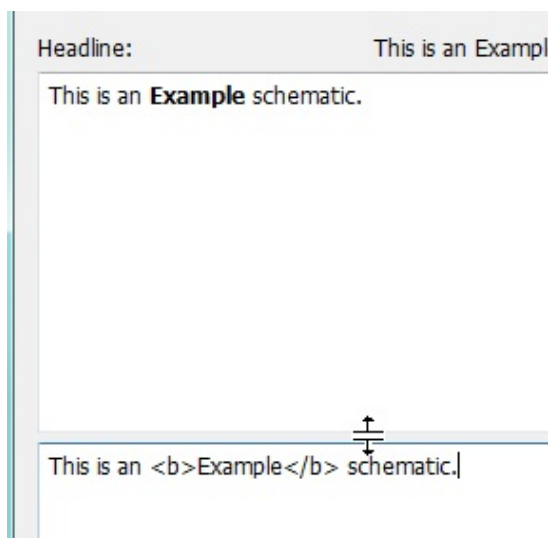


Now right click on the project and specify you are creating a schematic:

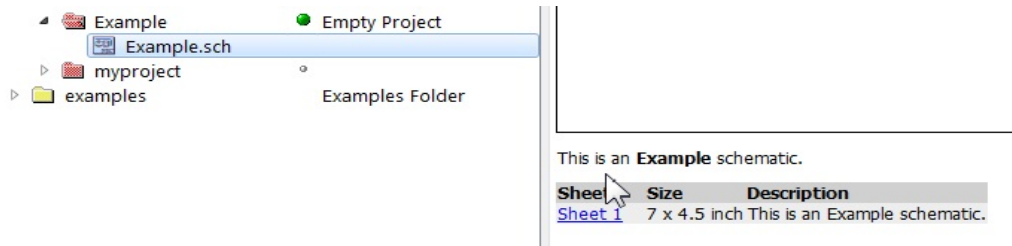


This brings up an empty schematic. At this point I immediately do a File | Save As to make sure the file has a reasonable file name. Since so far I do a single schematic/board per project, I just name the schematic the same as the project.

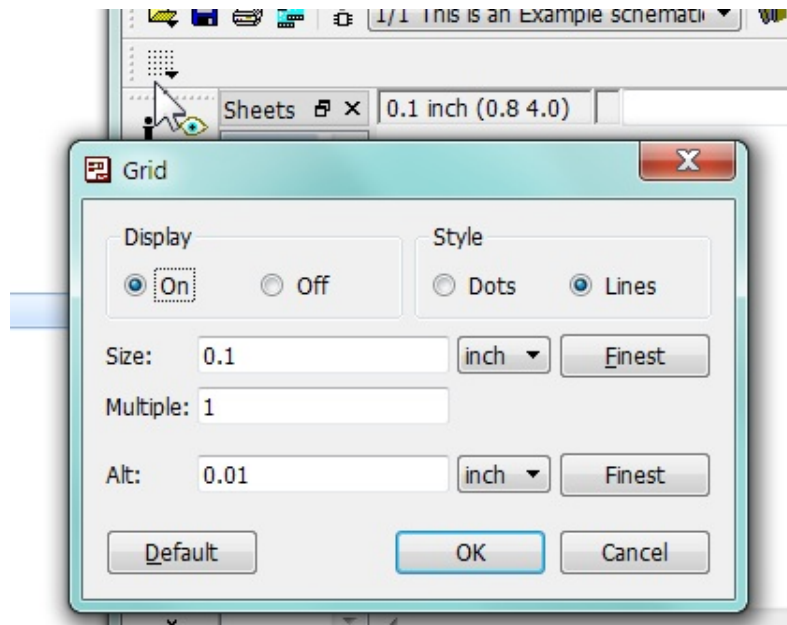
At this point you can do a Edit | Description and enter a description (using HTML tags if you want) of the schematic:



which will show up here:



Click on the grid icon (under the cursor here). Make sure the display is on and the size is .1:



Start Placing Symbols

Make sure all libraries are selected (or at least the ones you want). Click on Library | Use, then select the files (or all files) that you wish to use then click on the Open button. This is the c:\program files(x86)\eagle-6.4.0\lbr directory.

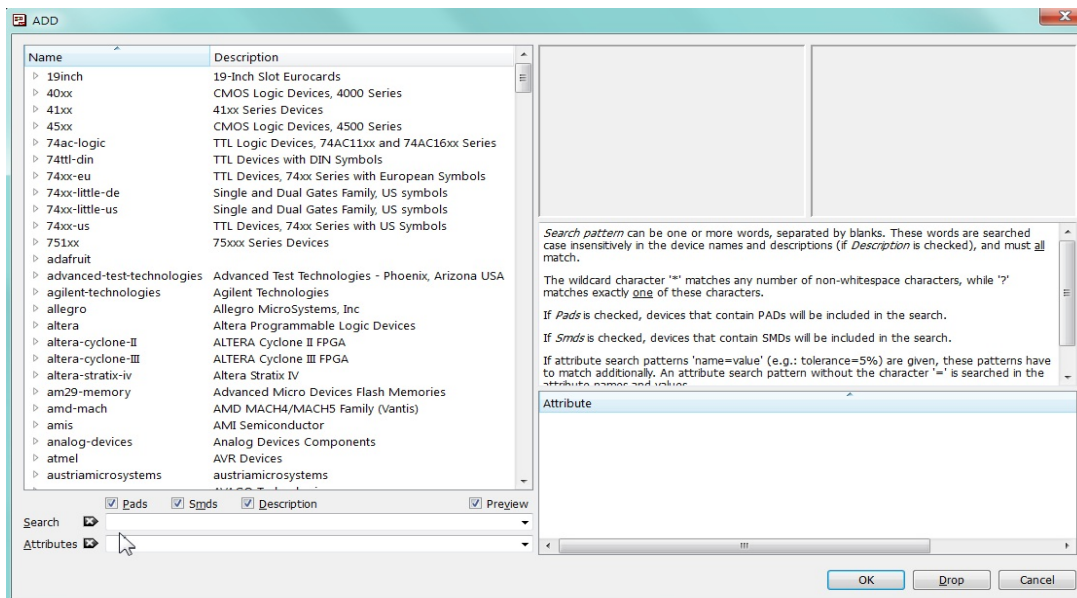
You are ready to start placing symbols. You will essentially find the symbol you want from the library and then paste it into the schematic.

Tips:

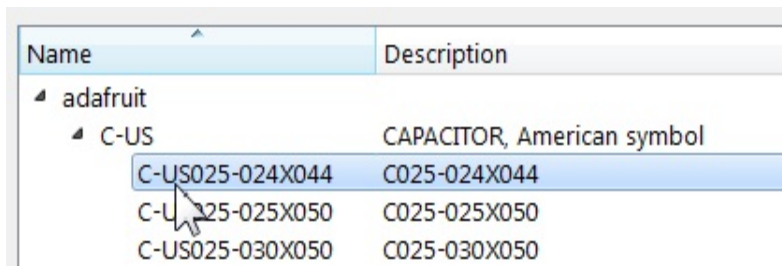
- Find the symbol that produces the correctly sized component on the PCB. Going back to fix sizing issues after the fact is pretty slow.
- Once I get the right symbol, I will add all of them at once rather than re-search the library for the same symbol.

To add a symbol:


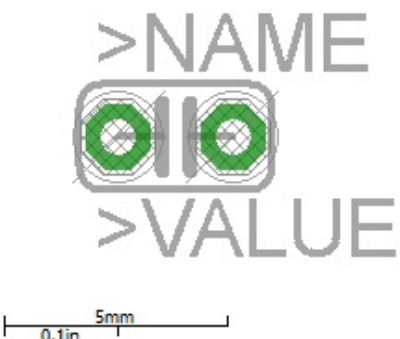
- Edit | Add and you will see:



- Type in a search phrase like 'capacitor'. This will display everything with the word capacitor in it. Note, I like the adafruit and RCL libraries - they seem to give the best results the fastest.


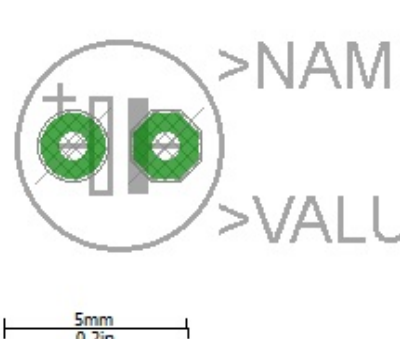


- Assuming you have the actual device at hand, look thru the symbols until you find one that matches the schematic symbol you want and has the PCB size you need. For example, here is one possible entry:

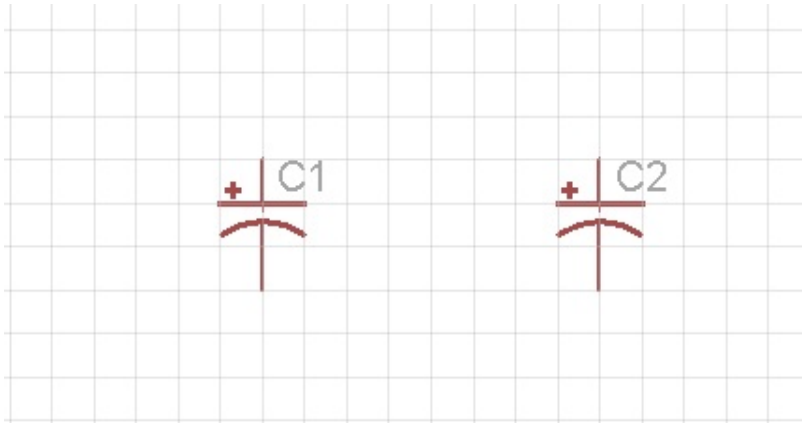
	
<p>CAPACITOR, American symbol</p> <p>Package: C025-024X044</p> <p>CAPACITOR</p> <p>grid 2.5 mm, outline 2.4 x 4.4 mm</p>	

This is unpolarized, using a 2.5mm grid and the outline is 2.4x4.4mm. You can see an approximation of the PCB format.

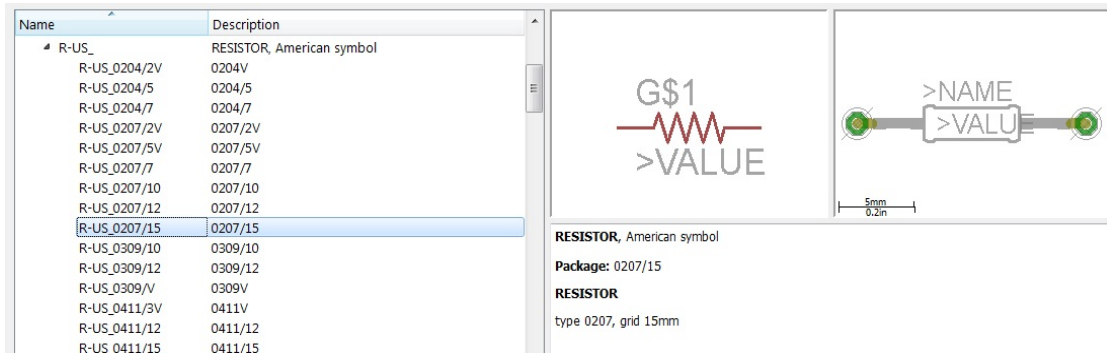
I really wanted a polarized capacitor so I keep looking thru the list until I find a schematic symbol I want and the size of the component matches what I really have:

	
<p>POLARIZED CAPACITOR, American symbol</p> <p>Package: E2,5-6</p> <p>ELECTROLYTIC CAPACITOR</p> <p>grid 2.54 mm, diameter 6 mm</p>	

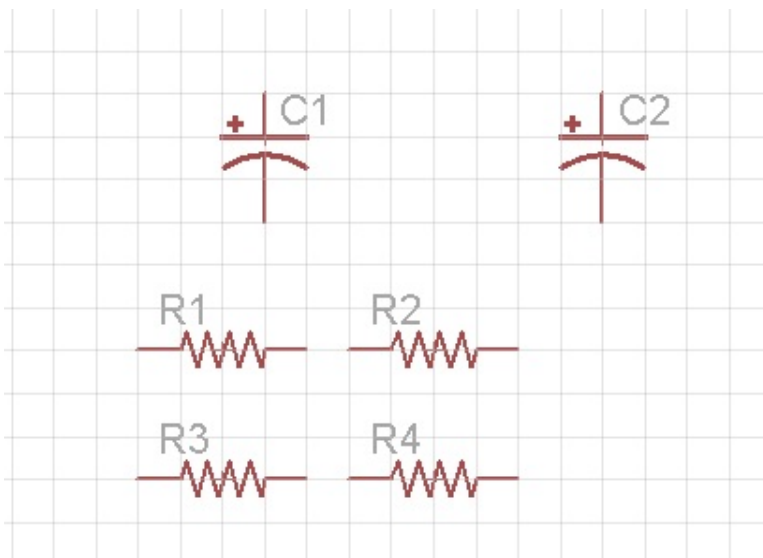
This is what I want so I click on OK and now I'm ready to click on the schematic 2 times because I need two caps like this:



I'm going to need 4 carbon resistors so I do another add, search for resistors, and look thru the library until I find exactly what I want:



Now I add 4 of those:

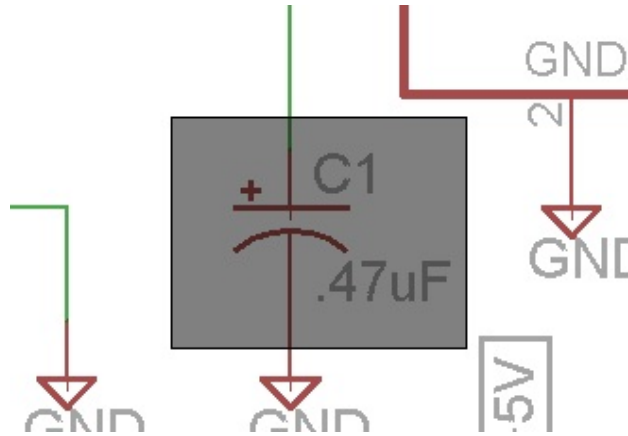


Stealing Symbols from Another Schematic

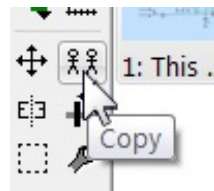
I frequently am going to use the same parts in various projects. Rather than sort thru a zillion components in a library it would be faster to just copy components from an old project.

To do this, have the new project and schematic open. Now double click on the eagle.epf file of the old project and then open the appropriate schematic there.

To copy a component, type ^g to go into group mode, then select the component:



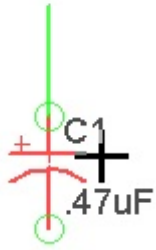
click on the copy button:



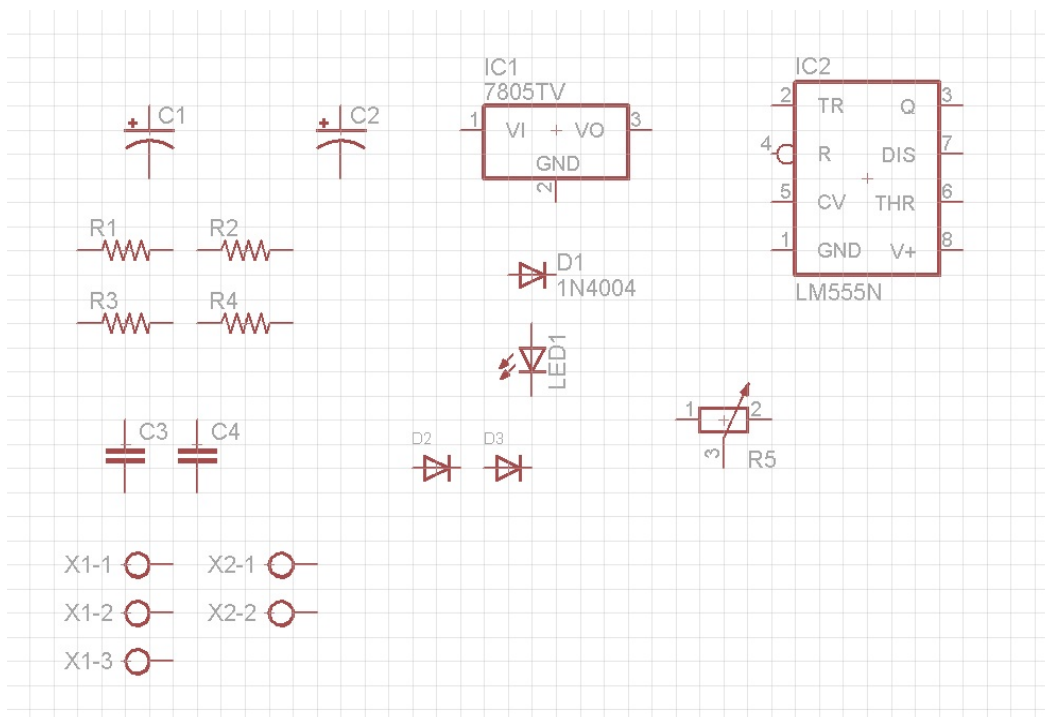
now go to new schematic and press the past button:



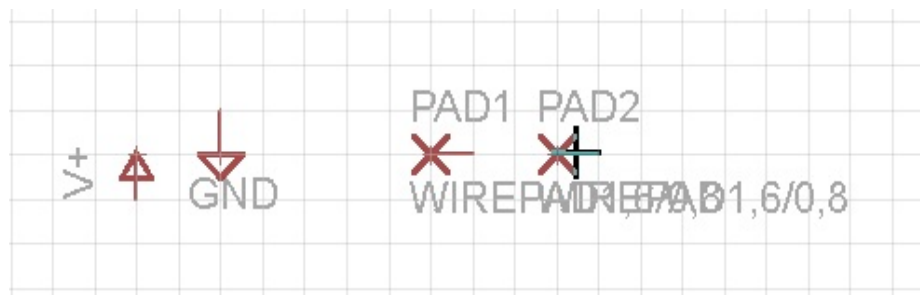
and now place the component:



I've about got everything here, but I will also need to add V+, GND, and some pads for the off board switch to connect to:



and here are the last parts:



Lay Out the Parts

Now that I believe I have all of the parts I can lay them out. Typically input is on left, output is on right.

Moving components is one of the non-standard parts to Eagle. You can't just drag and drop or select. You must first do a 'move command', then you can click on a component and move it.

There are 4 ways to indicate you want to move: ctrl-M, type 'move', click the move button, or right click on the component you want to move. The last tends to be the easiest.

Start moving stuff to where you want it. As you move, the right button will rotate the component that is selected.

To zoom in/out use the mouse scroll wheel. To pan, click on the scroll wheel and drag. Like sketchup.

You are going to want to move groups of components. This is really bizarre to handle.

First select the components. You can either type ctrl-g, group, or click the group button. The 1st tends to be easiest. Now you can drag a box around the components to select.

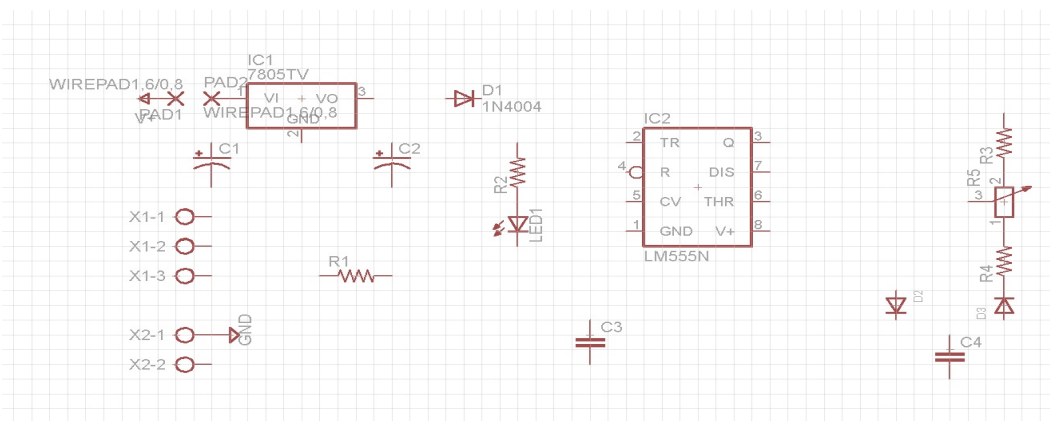
If you drag the box from right to left, anything the box touches is completely selected. If you drag the box from left to right, then only those things completely enclosed are selected.

Once the components are selected you have to go into move mode. Sometimes this is done automatically, sometimes not, so I just type ctrl-M to guarantee I'm in move mode.

Now hold down control and use the right mouse button to grab a component in the group and you will be able to move them all.

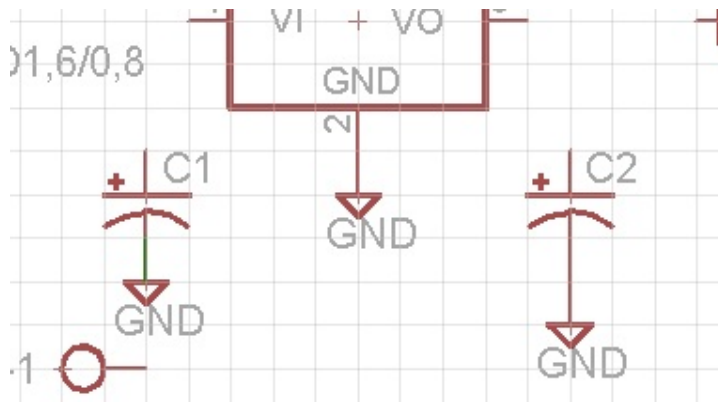
It doesn't get much more convoluted then that!

After moving and rotating all of the components, I have most everything roughly where I expect it to be:



Before wiring up, copy V+ and GND and place those where components directly connect so the

wiring is as clean as possible:

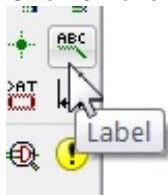


In the case of this design, V+ only connects to the on/off switch, so I can't use it else where, but I could build my own label like this:

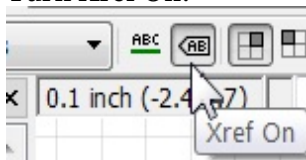
- You have to draw a wire to the pin that will be labeled:



- Click on the label button:



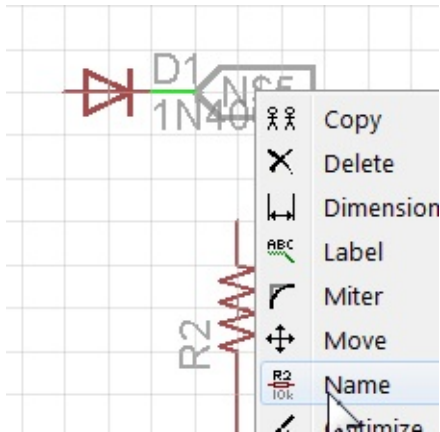
- Turn Xref On:



- Click on the wire and place the label:



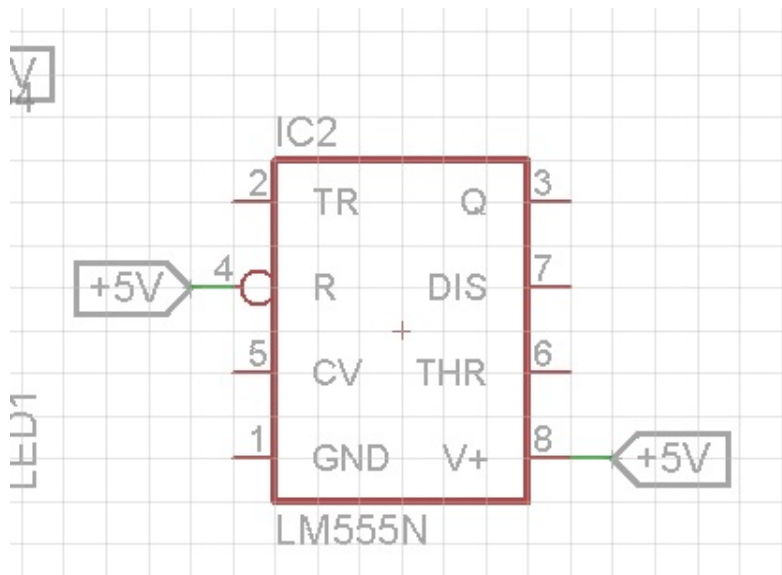
- right click and choose name:



- give it a reasonable name:

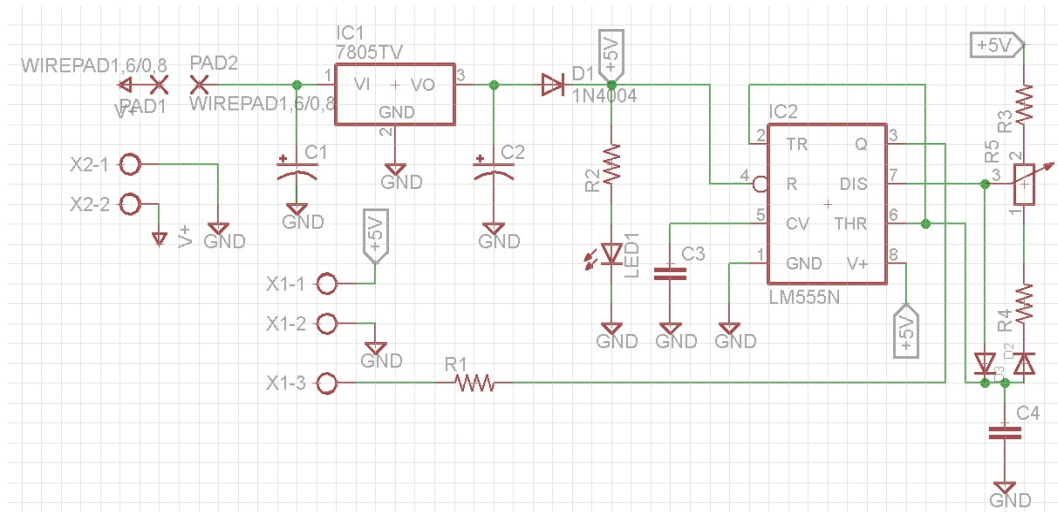


- and now you can connect other component pins to that name using the label command in the same manner which, in essence, ties all of the pins to a single wire:



At this point you should be ready to start wiring components. Wire everything, move components as necessary and get a reasonably clean schematic:

This should be pretty close to laid out properly:



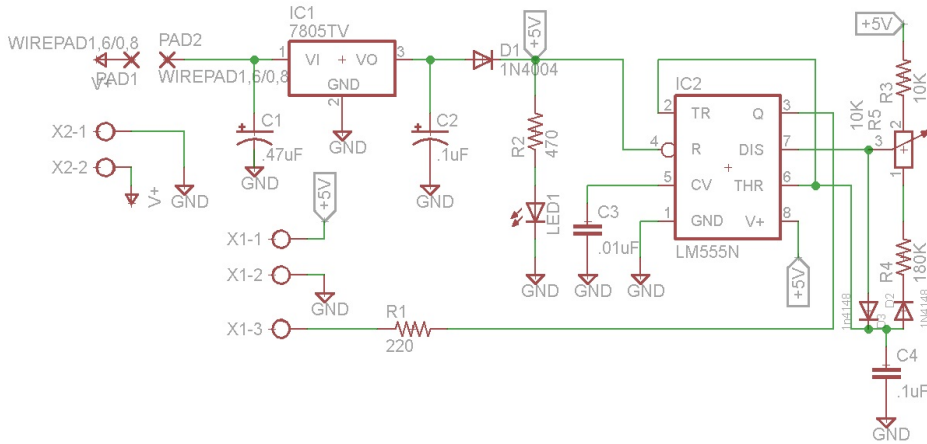
Now you can enter values for the various components. Right click a component and select value:



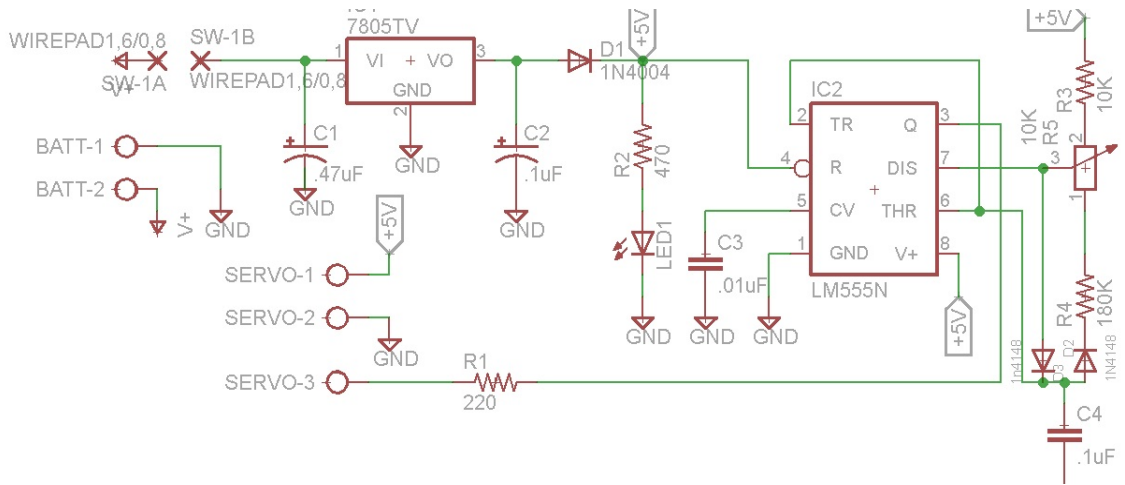
The value is now displayed:



Click on each component and the value box will pop up so you can fix them all:

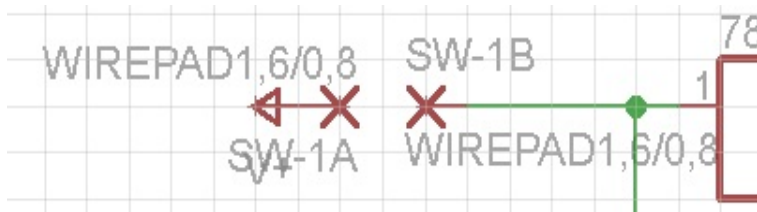


Some components like the pads and the connectors don't have values, but I can change their names so they make more sense:

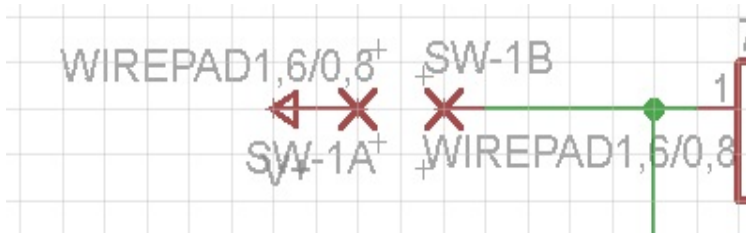


Everything is now named. Now, move things around a bit more so names are all clear.

This is quite a mess:



To fix this, smash the pads (click smash button, then click on the 2 pads). This separates the pads into the actual graphic and the text. The text has little + handles:



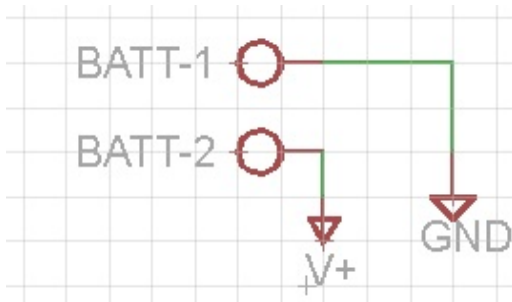
I want to get rid of 'wirepad', so I press `ctl-del` to go into delete mode, then click on the + next to those words. Then I moved the sw-1 text blocks to make something clear:



The V+ here is rotate wrong:

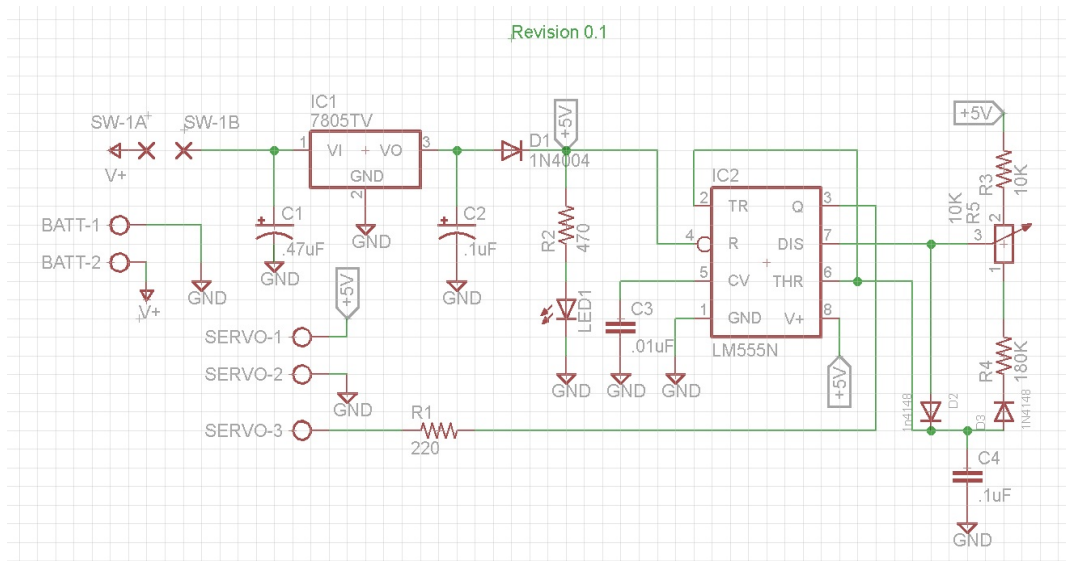


I smash it and rotate just the text:



note that pressing `ALT` while moving allows precise alignment.

I use the text command to put in revision information, and I have what I think is a completed schematic:



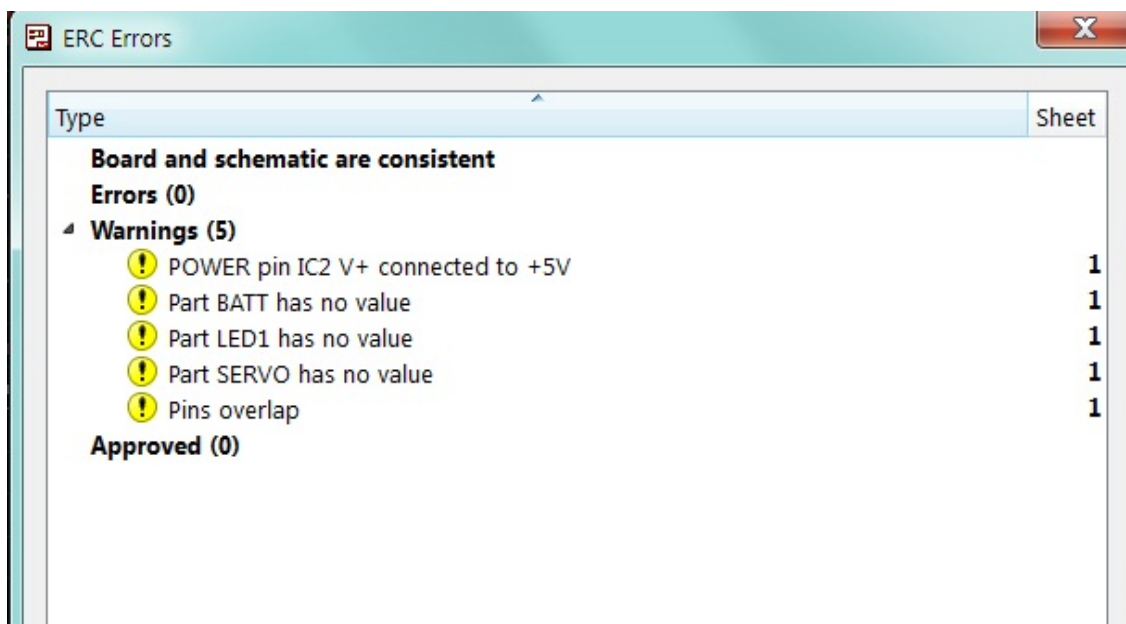
Finally, you can go to Tools | Renumber Parts if you added/deleted components and the numbering (R1, R2, ...) has holes.

Rule Check the Schematic

Once you are done entering your schematic, it needs to be checked. Press the ERC (electrical rule check) button:

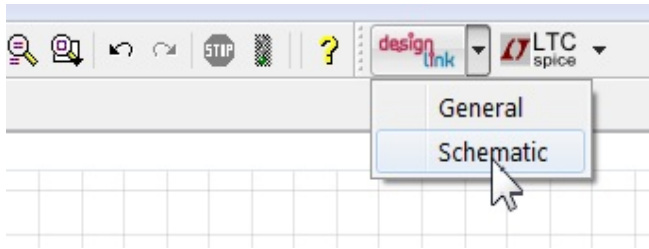


here is what I got:



correcting problems is outside of the scope of this procedure. See the manual.

You could press the design link + schematic button at this point:



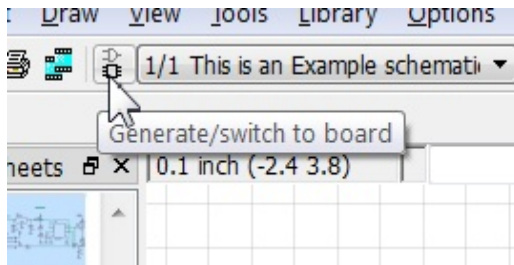
which will allow you to build a shopping list. Since it is for Newark, I'm not overly interested at the moment.

You can also export | partlist to generate a nice TXT file of the parts.

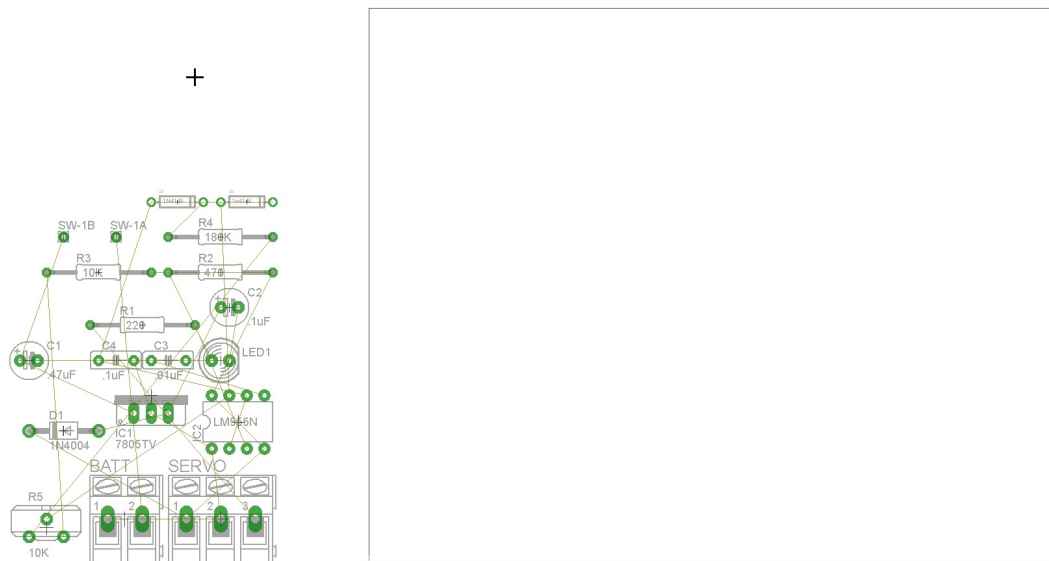
You can export | netlist to get the list of nets to check your schematic.

Laying out the Printed Circuit Board

I am ready to layout the PC Board. Press on the generate/switch to board button:



This goes to board view, creates an empty board, and puts all of the components to the side and connects everything with 'airwires':



The board is drawn at the max size possible (in the free version anyway). Everything must fit into this space.

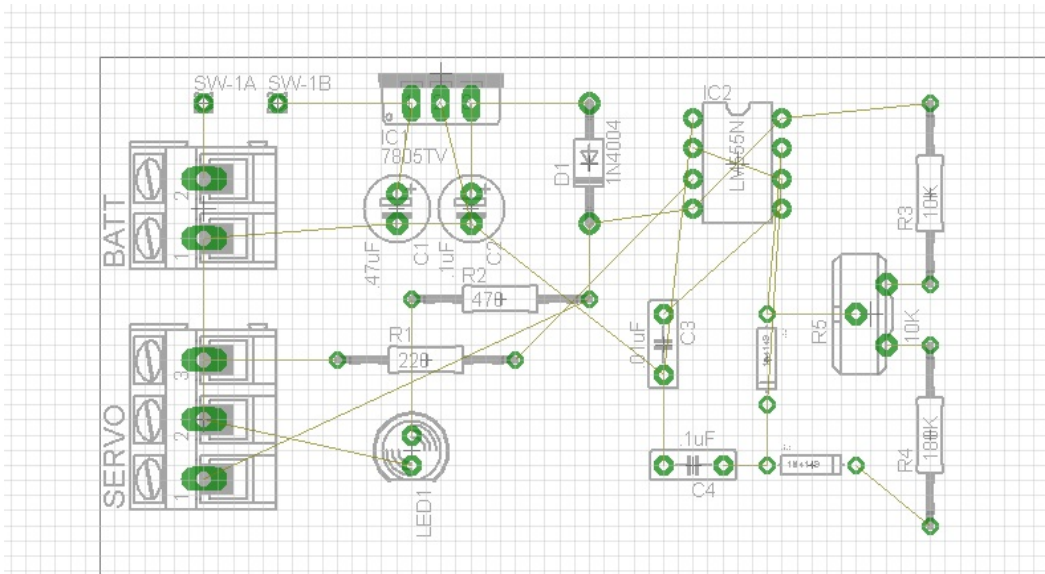
Enable the grid. The default of 50mils is probably fine. If you are just trying to layout components onto perf board, use the hole spacing of the perf board (probably .1").

Start moving components onto the board in the manner in which you want them. Move/rotate works the same way.

As you are placing components, you can click on the ratsnest button for the program to rewire the components based on how close the components are as you move them. This helps visualize where the actual traces may go.

Then just work out a layout that is reasonable. Probably want to leave some space around IC's since many traces will come in.

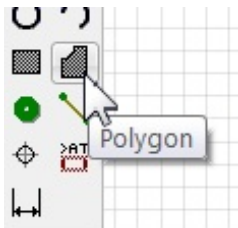
Here is my first attempt:



To generate actual copper traces, first click on Edit | net classes and change default width, drill, and clearance to 30mil/20mil/30mil.

DO NOT use a width < 30 except in special cases. I've had too many problems with skinny traces tearing.

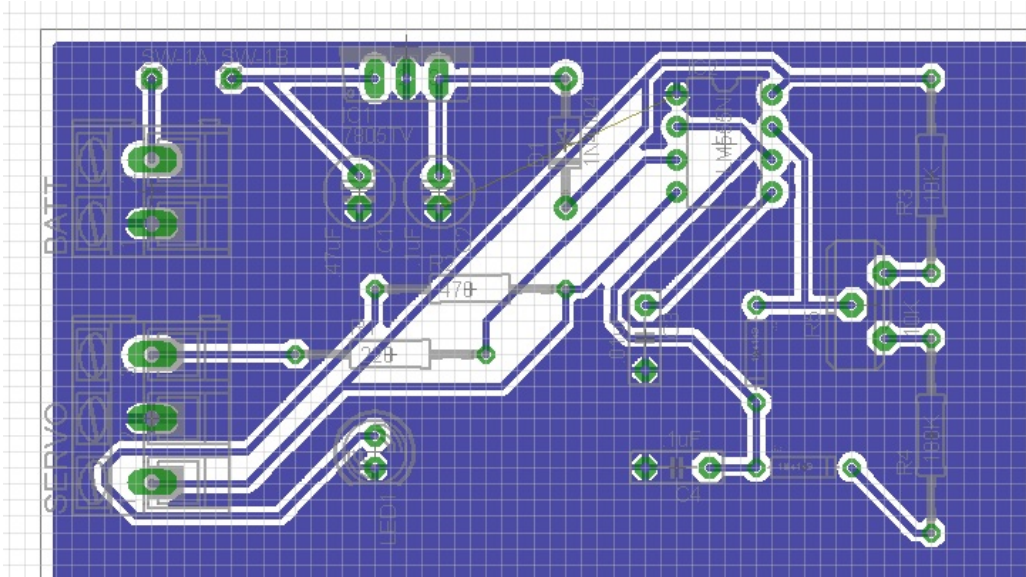
Next click on the polygon tool:



make sure the layer is 16-bottom (blue) and draw a polygon around the outside edge of the board. Right click the dotted blue line and for name enter GND. Click ratsnest to fill in the polygon (if the polygon is acting weird, use ratsnest to refill it). This creates ground in all unused places on the board. This large ground area prevents noise, so it must be done.

Click on the auto router button. Top should be N/A. Bottom should be the preferred direction for traces (I'm using *).

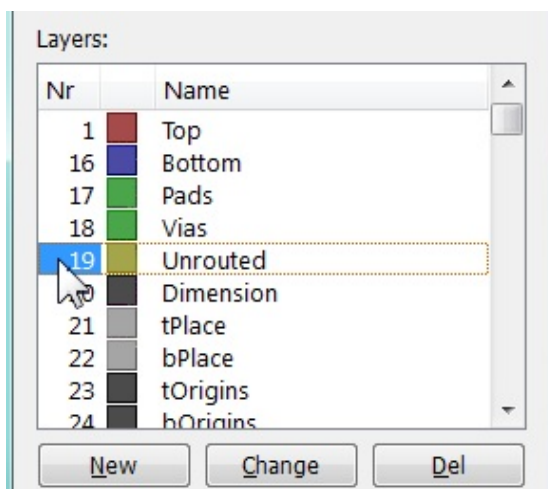
you will get a routed board like this:



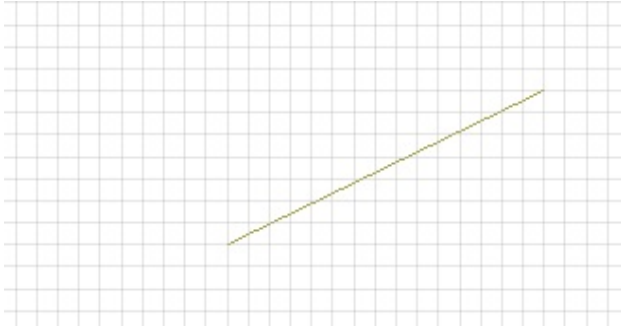
Look at the status line to see if the board successfully routed. In this case, it is only 96.8% complete:

Optimize4: 96.8% finished. Polygons may have fallen apart.

That means there are still airwires and one or more connections were not made. It is rather hard to see the airwires with the blue background, so I turn off all layers except unrouted:



and this lets me see where airwires are left:



now turn all layers back on and you should see where the airwire is (pointed to by red arrow):

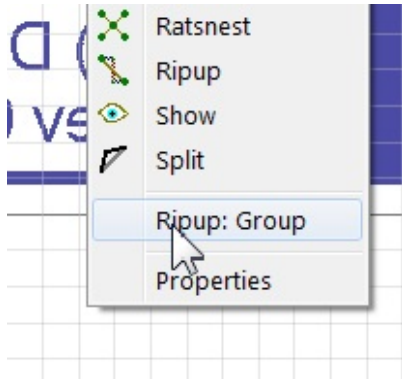


The typical reason I've had trouble is an area of ground is isolated from the other grounds. In this case there is an 'island' of ground pointed by the green arrow. This island needs to be connected via a jumper to another ground (the jumper will run on the top of the board). We will address that problem in a moment.

Look at the traces and make sure you are happy with them. If you want to rip up the board after you put in jumpers, you will have to redo the jumpers, so do that last.

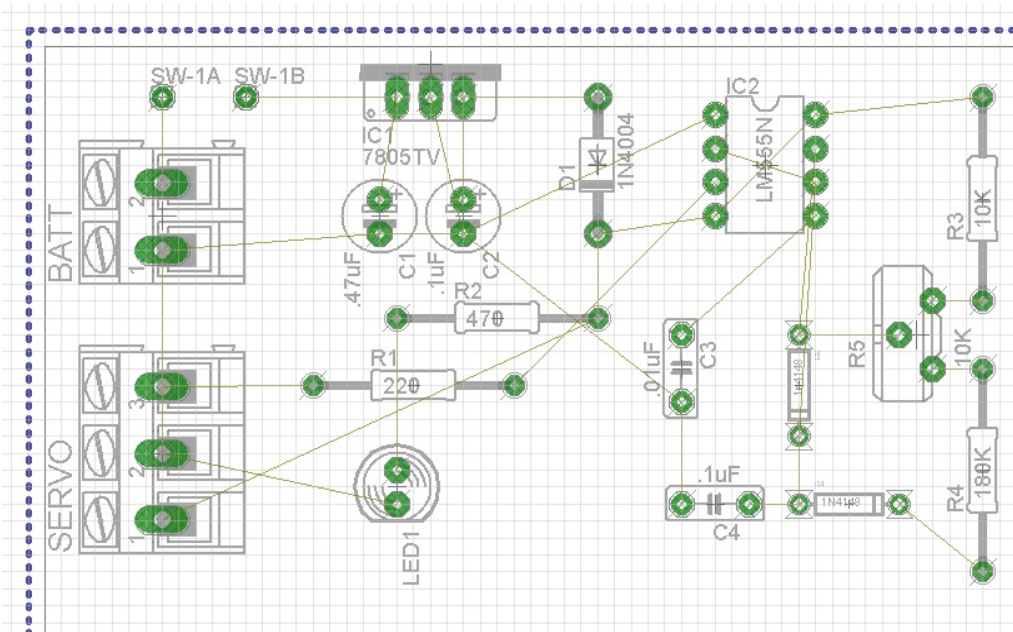
To rip out a single trace, simply right click on it and click on ripup.

To ripout everything, select the entire board (group). Right click on one of the traces and select ripup. That only removes one trace, but now right click on another trace and you get the ripup group option:



select that and all of the traces are removed.

This will get you back to where you started:

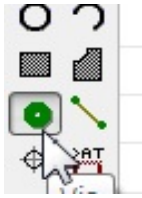


You can now move components and do the route again.

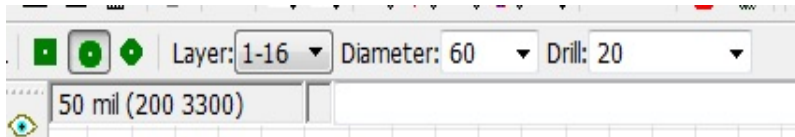
Fix Ground Islands

To fix ground islands, we need to put a via in the island and a via somewhere that is properly grounded. These will be connected by a jumper wire (actual wire) on the top of the board since I am not attempting 2 sided boards yet.

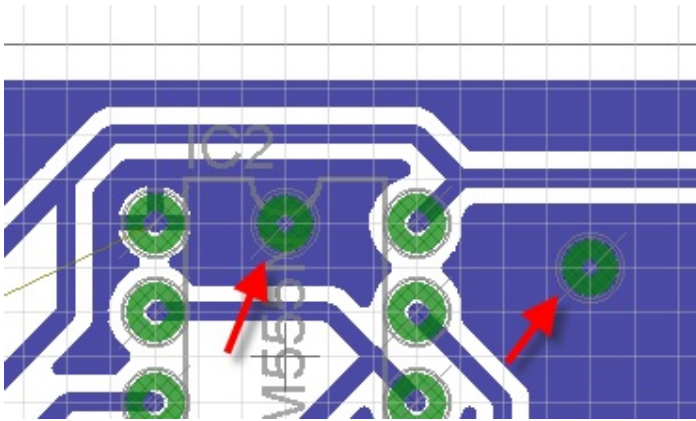
Click on the via button:



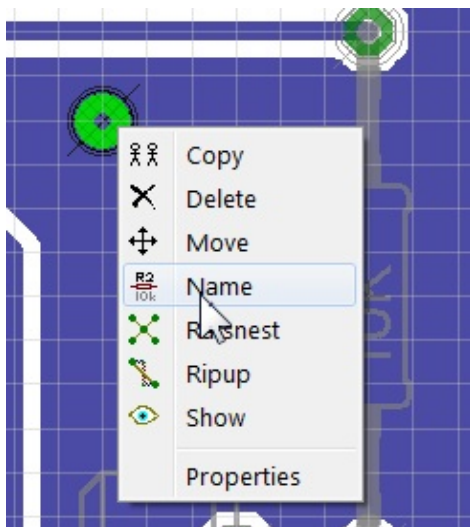
Make the pad round, 60 mils in diam with a 20mil drill hole:



Now place the vias. One in the island and one nearby in grounded space:

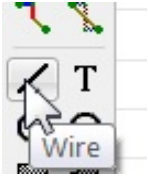


Right click each via and change the name to GND:

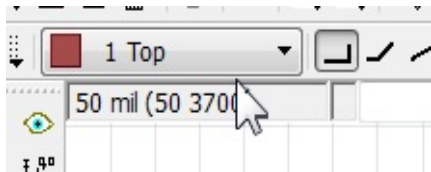


when I click ratsnest now, the old airwire is replaced with one between the two vias.

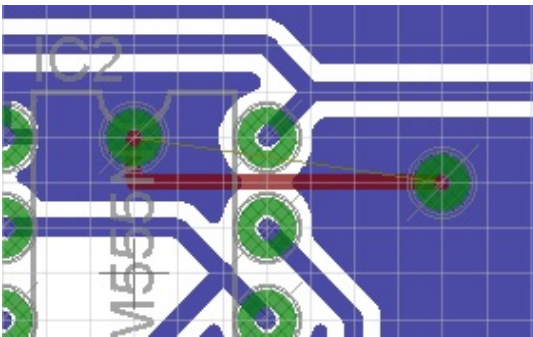
Now click the wire tool:



and make sure the top layer is selected:



and draw a wire between the two vias:

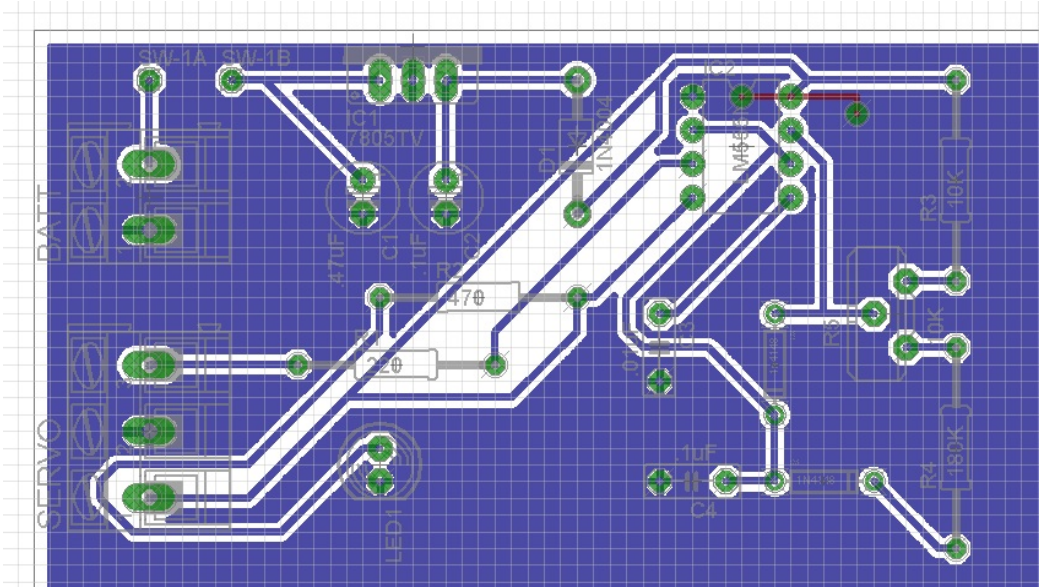


Right click the wire and check the name - it must be GND.

Click ratsnest again and there should be no airwires left.

Click autoroute again and it probably will resolve the problem to your satisfaction (if it was close to perfect the first time).

I'm getting close to done:

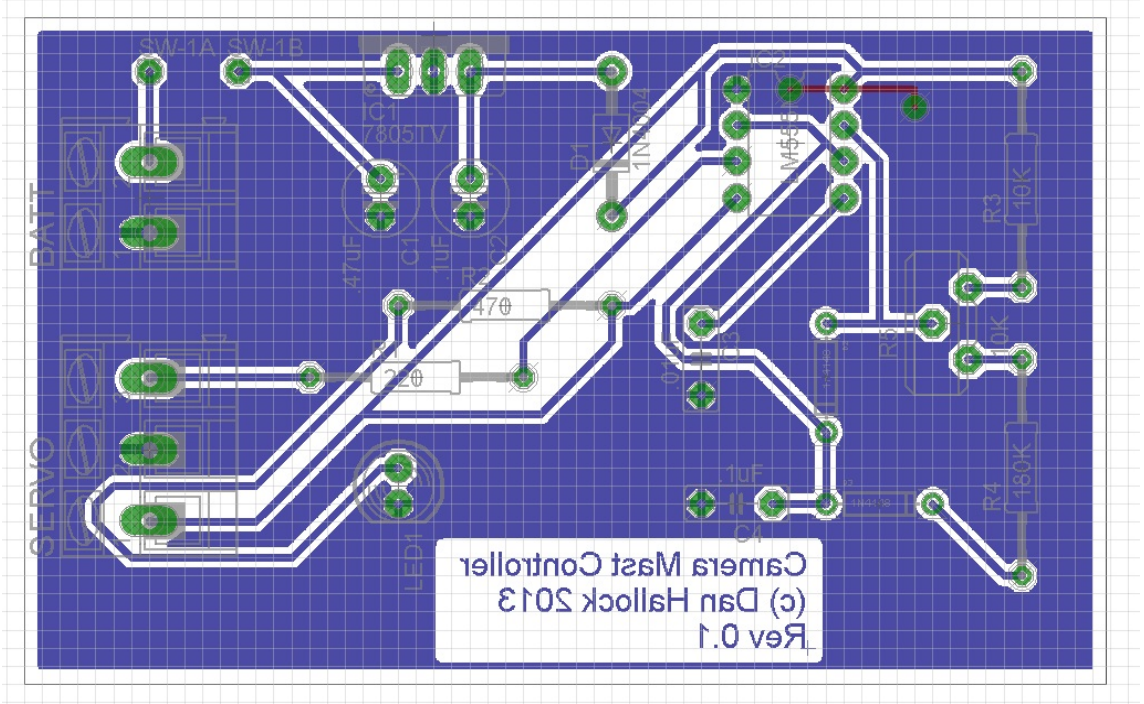


Putting Text on the Board

Now I want to write some board information into the bottom copper. Click on the text button. Enter your text. Before placing it, select the bottom layer and perhaps size information.

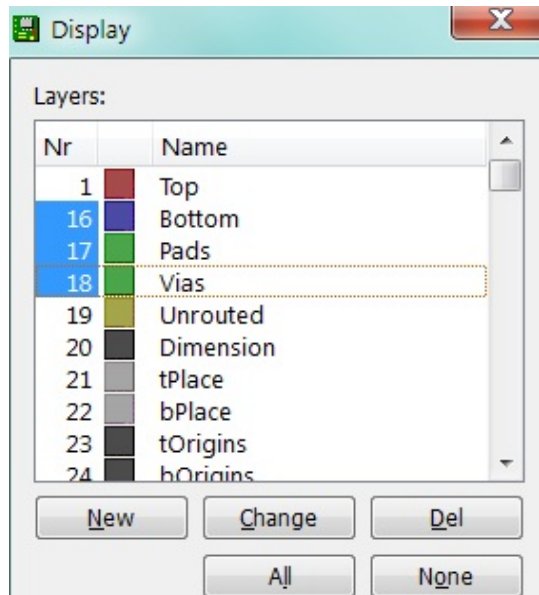
Click on where you want to place the text. NOTHING HAPPENS. That's OK. Now press ratsnest again and the text will appear. It should be backwards since you are looking at the top of the board, not the bottom.

Finally, if you are happy with the overall layout, move the edges of the board in to the final board size. Click on rats nest again to clean it up and you should have your final board:



Printing the Printed Circuit Board

Go to the layers settings and select only bottom, pads, vias:



Type **ctl-P**, make sure the correct printer is setup and is setup for b&w only. Select the black and solid options and make sure the scale factor is 1.

Check each component and verify it fits - its too late once you etch the copper and start drilling.

Save everything! When you come back into the board view, the ground polygon will be missing. Just click the ratsnest button to bring it back.

Printing Double-sided PCB and Silk Screen

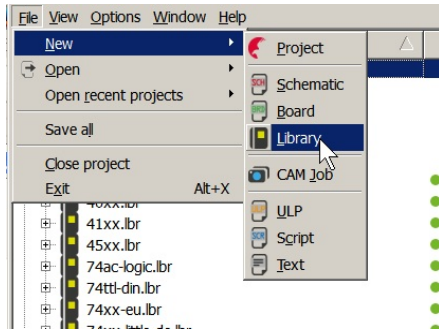
I covered this in my blog:

<https://bigdanzblog.wordpress.com/2014/12/16/printing-double-sided-pcb-and-creating-silk-screen-from-laser-printer-toner/>

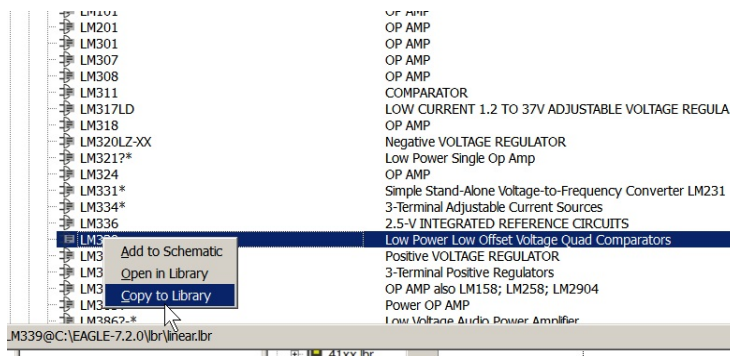
Creating A Component

The best way to do this is find an existing component close to what I need. I am going to copy that device to an empty library (because it could have lots of extra stuff I'm going to delete and this makes it easy to track down the extra stuff), edit the component in the temp library, and finally copy it to the destination library.

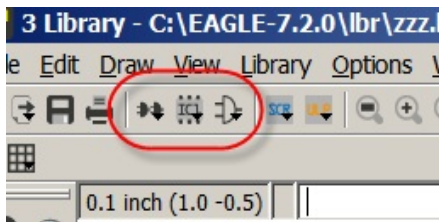
- Once a suitable component is found to copy, open a new library:



- The new library is called 'untitled'. I save it as zzz.lbr.
- In the eagle control panel, locate the source component to copy and copy it to the open zzz library:



- Use these buttons to see the various parts of the component so you can edit them:



- Once done editing the component to your satisfaction, copy it from the temp library to the final library as shown above.
- Delete the temp library.

Random Notes

- To hide all GND airwires, right click on one, then select hide airwires.
- To bring back the GND airwires, type 'ratsnest gnd'
- Exporting Libraries

Once a project is about done (at least as far as all components are selected), you can export all of the libraries used into a single directory. You can then use this library and not have to worry about changes in the main library.

This will copy the entire library. So if you use just one capacitor, you will get the entire library of capacitors.