

VeeCad Notes

These notes cover how to use VeeCad with an EagleCad generated schematic. VeeCad is a stripboard (strip board) / Veloboard editor which allows you to lay out such boards.

Contents

Create the Schematic.....	1	Layout the Components.	8
Export Net List.....	1	Building the Strip Board.....	8
Import Net List.....	2	Test the Board.	11
Defining New Components.....	4		
Select Board Size.	7		

Create the Schematic

Create the schematic for your project in Eagle as usual. Once the schematic is done and passes ERC, you are ready to export the netlist to VeeCad.

Export Net List

You are going to need the netlist_protel User Language program for EagleCad. This is found here:

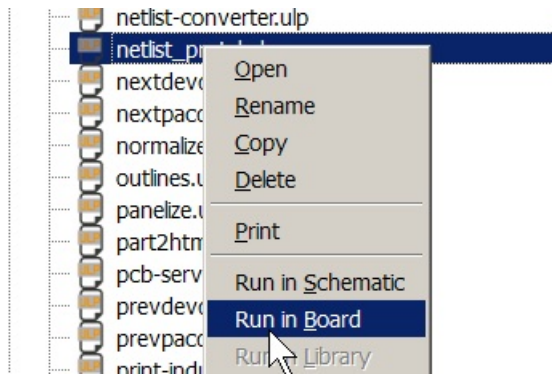
ftp://ftp.cadsoft.de/eagle/userfiles/ulp/netlist_protel.ulp

Move this file to the Eagle ULP directory.

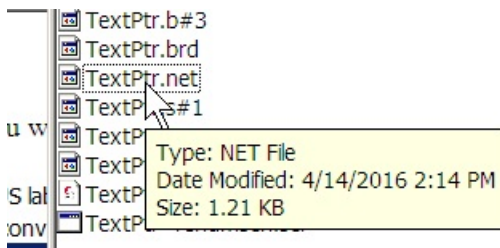
In the Eagle 'control panel', go to User Language Programs and you will find 'netlist_protel.ulp':



Right click and run in board (**don't use schematic** - it will cause import errors):



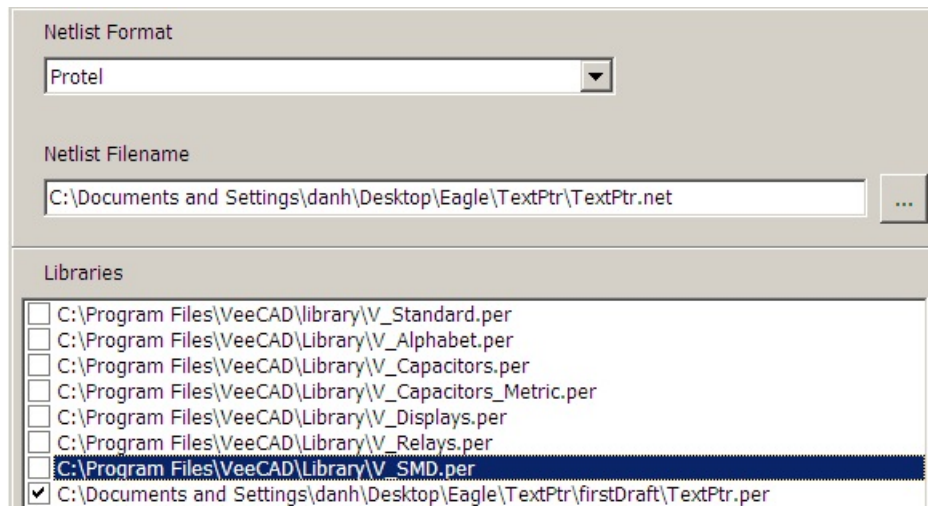
Now there will be a .net file:



Import Net List

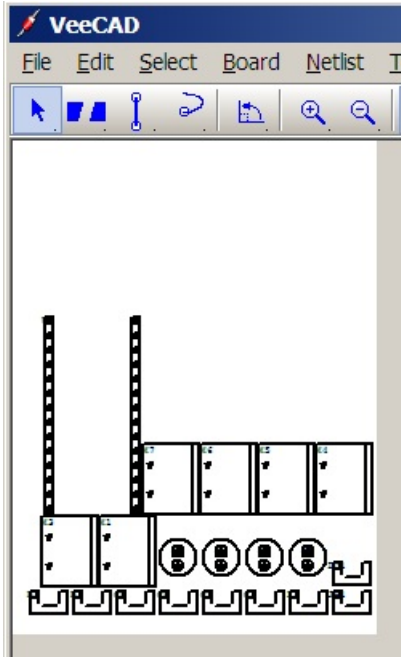
Start VeeCad, and click on Netlist | Import.

This will allow you to specify the .net file you just created. such as:



The standard libs appear to be useless because they don't work with Eagle; however, you can specify past projects. Here I specify my prior attempt at txtptr and it pulls out many of the last component outlines.

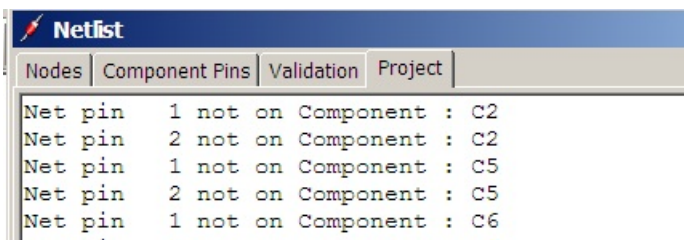
So I end up with this:



It did a decent job as I had already designed several of the components in the prior draft. All of the U shaped outlines still need to be defined.

Now go to Netlist | View | Validation. This should be clean or at least the errors make sense. If not, you will have to fix .net and import again.

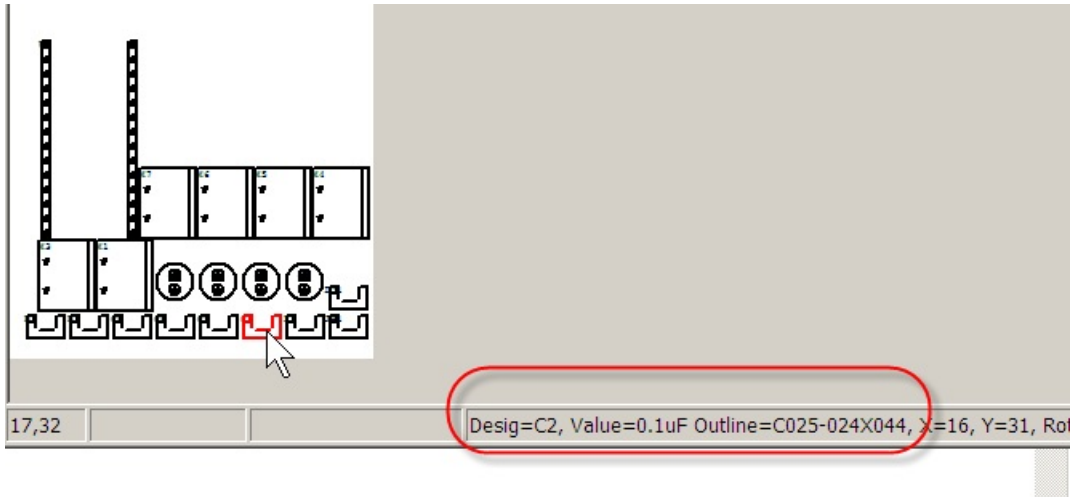
Look at the Project View. Chances are there will be errors which are likely due to the component not yet being defined:



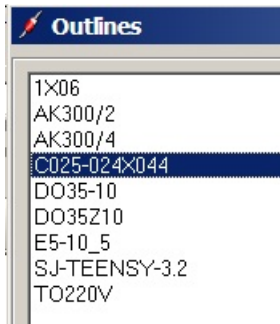
Defining New Components

Fortunately this is pretty easy. C2 is a small ceramic cap. We define an outline, then assign C2 to that outline. Note that the pins for C2 are 1 and 2. since this is not polarized, we don't have to worry about that yet.

I find C2 and once selected I can see I have C2 and also see the outline name:

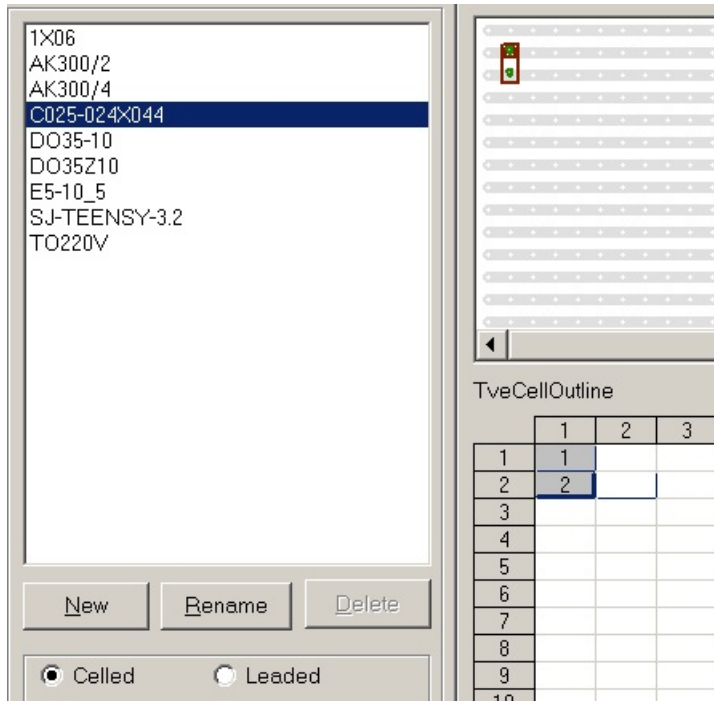


I need to define outline C025-024.... . Now click on Board | Outlines and select that outline:

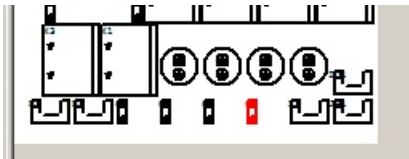


Cell Outlines

In the outline array right click to remove blocks and left click to add. My little capacitor is very simple and I label the pins 1 and 2:



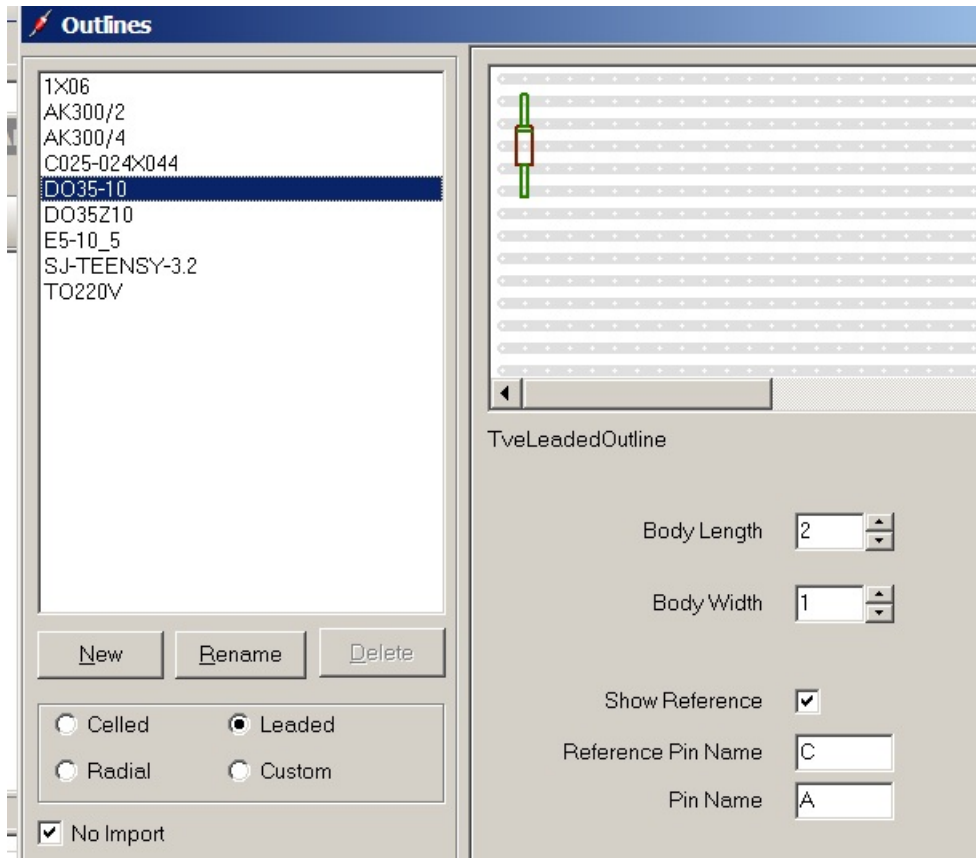
click on OK and now all capacitors using that outline have been corrected:



and now if you Netlist | View | Project, those caps are clean.

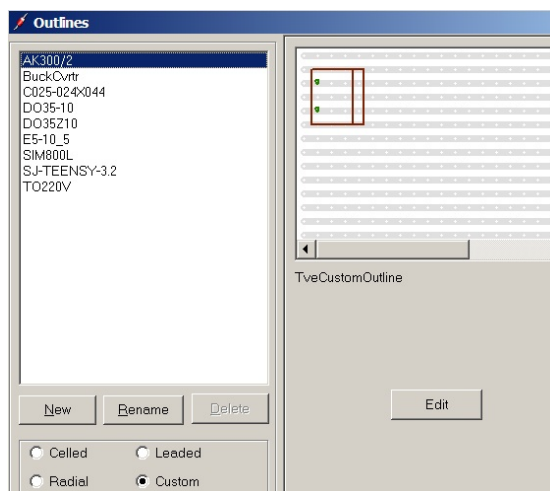
Leaded Outlines

D1 is going to be a leaded outline setup like this:

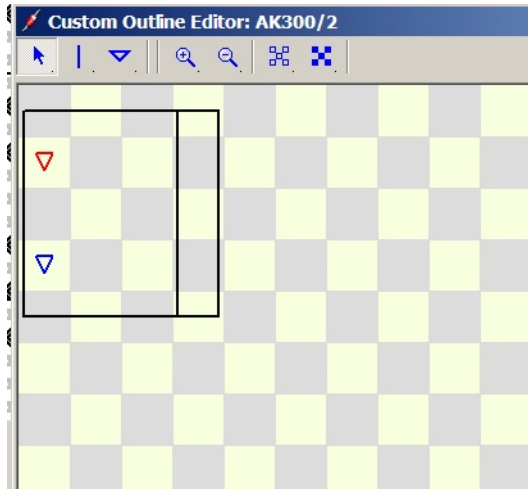


Custom Outlines

Although the AK300/2 (a terminal block) outline imported from my TextPtr library, it is a good example of a custom component. Here is the basic definition:

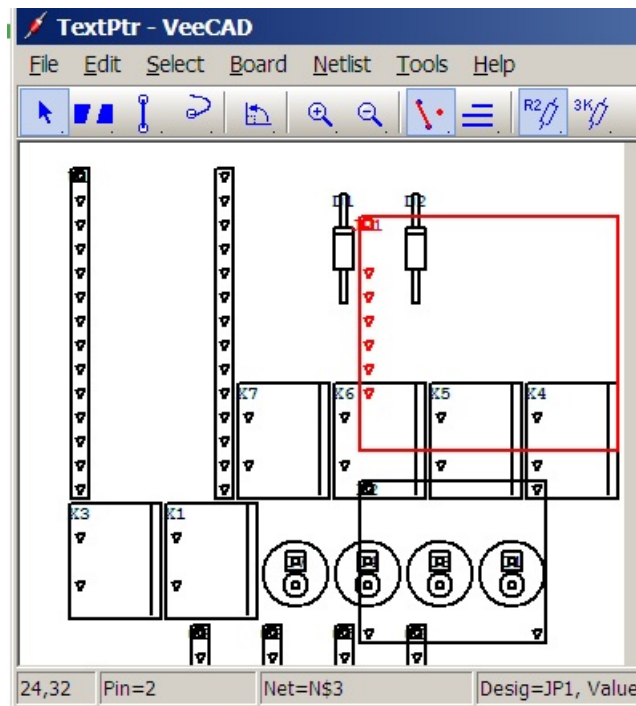


Click on the edit button and you will see:



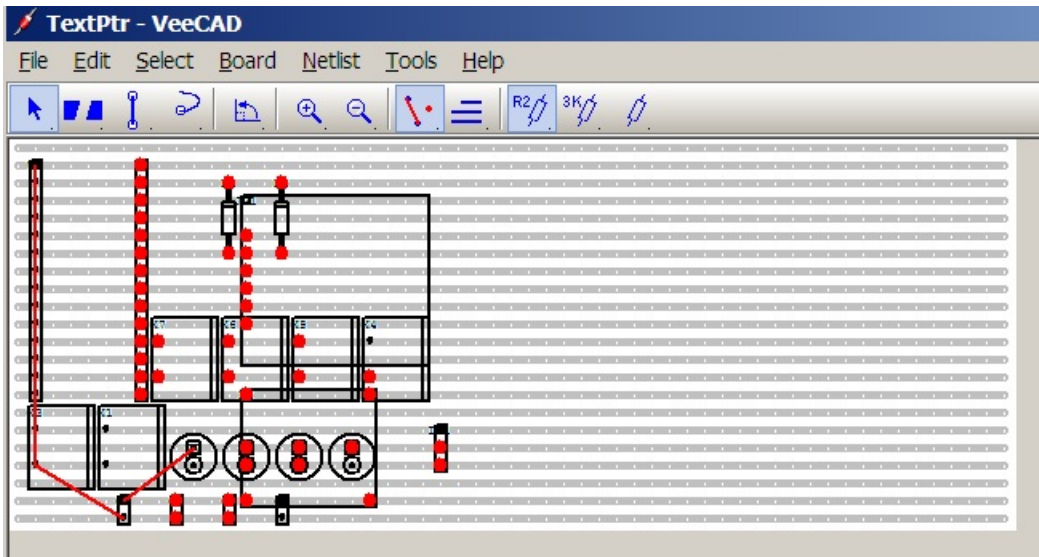
Here you use the line tool to create lines, and the triangle tool to create pins and assign them names. The process is pretty quick and easy.

Continue until all components are defined (no U shaped components):



Select Board Size

Now select the appropriate board size. Don't forget to verify the board is going to fit. Changing the size after the fact can be nasty.

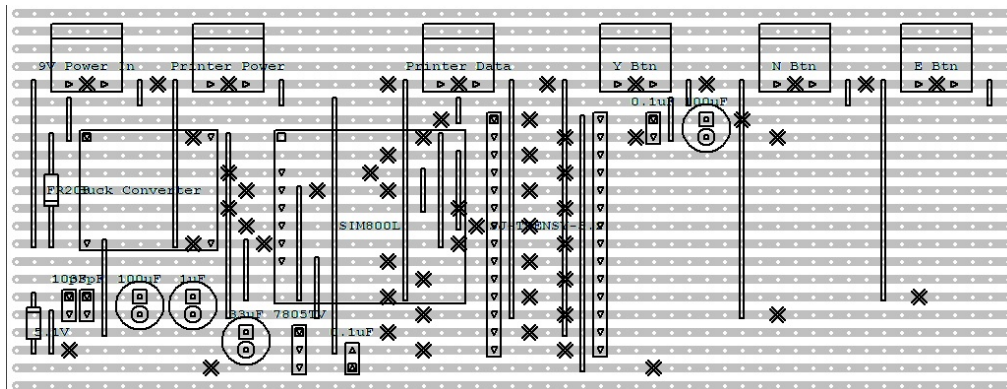


Layout the Components

You are ready to go!

Don't forget ALT-DRAG lengthens wires and leaved components

Once the board is layed out it will look something like this:



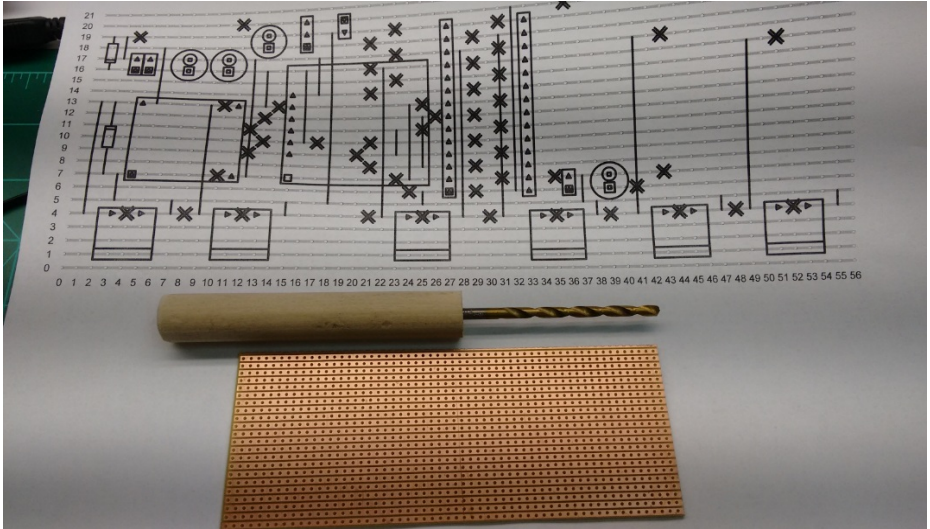
Building the Strip Board

Once I'm ready to build the board, I print the front diagram (shown) and the back. You will need to use the back to layout the holes.

I built my trace cutter using

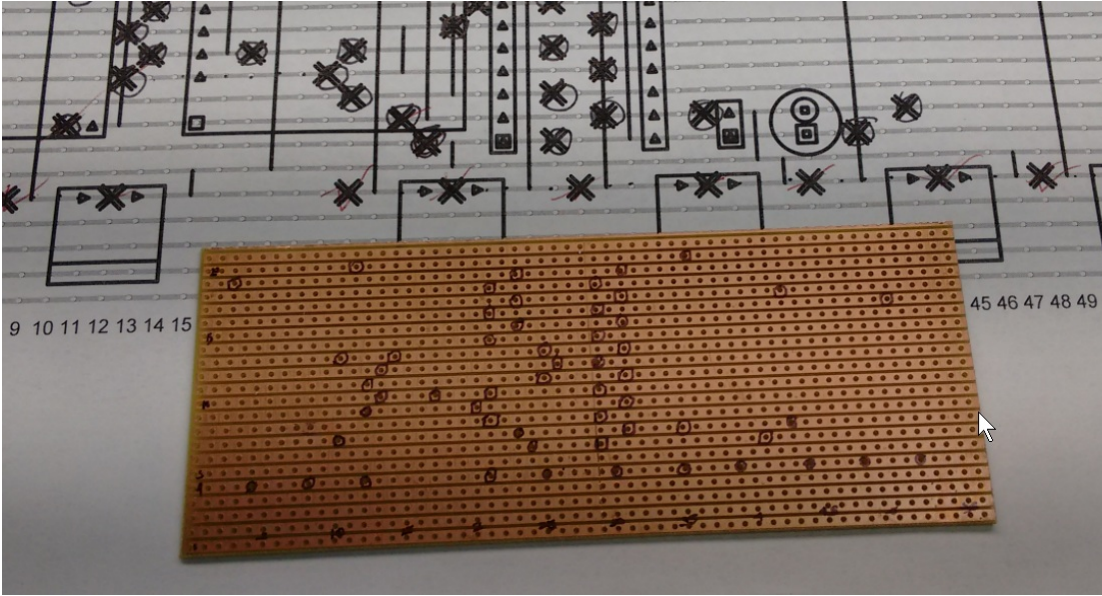
<http://www.instructables.com/id/Stripboard-track-cutter/>

Then the PCB is cut to the necessary size.

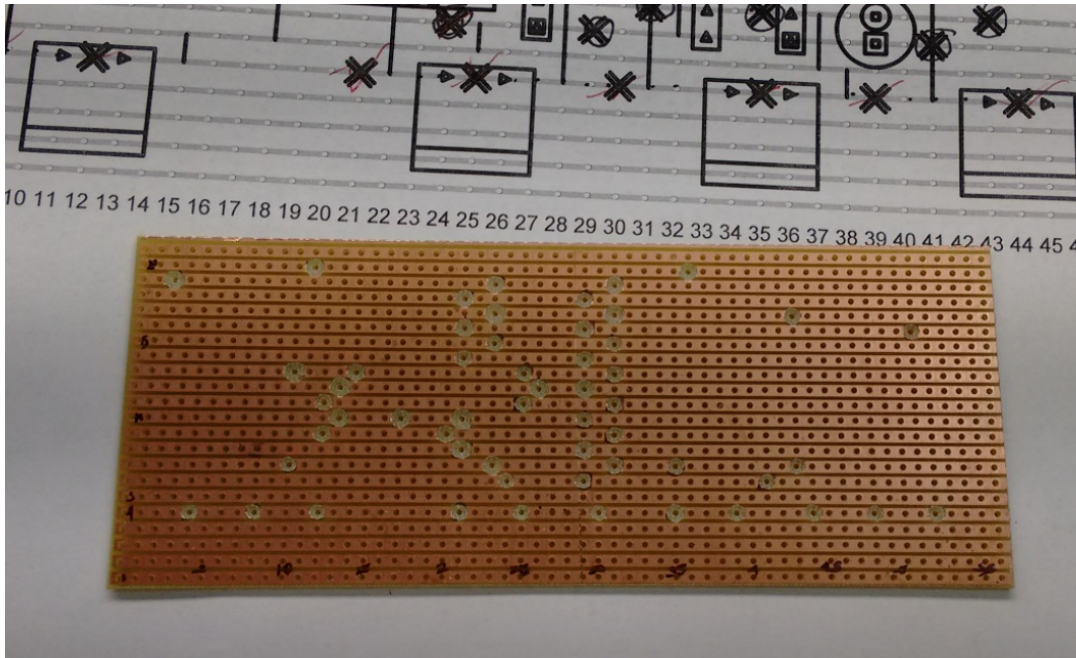


Cutting Traces

Next the X's are laid out on the board. I numbered the board by 5's to match the document. I then used the edge of 2 sheets of paper to work my way up the board row by row, marking each hole:

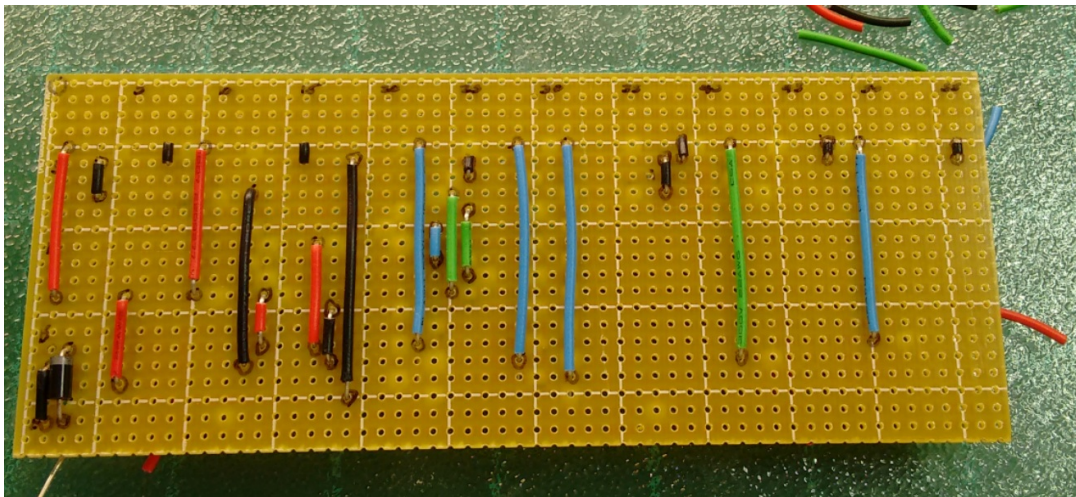


Now using the special tool, cut out all of the circles:

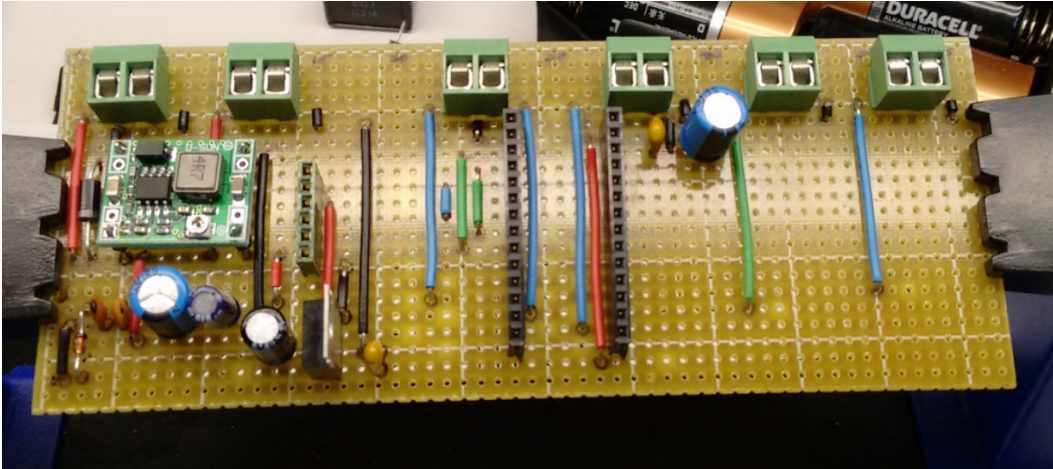


Installing and Soldering Components

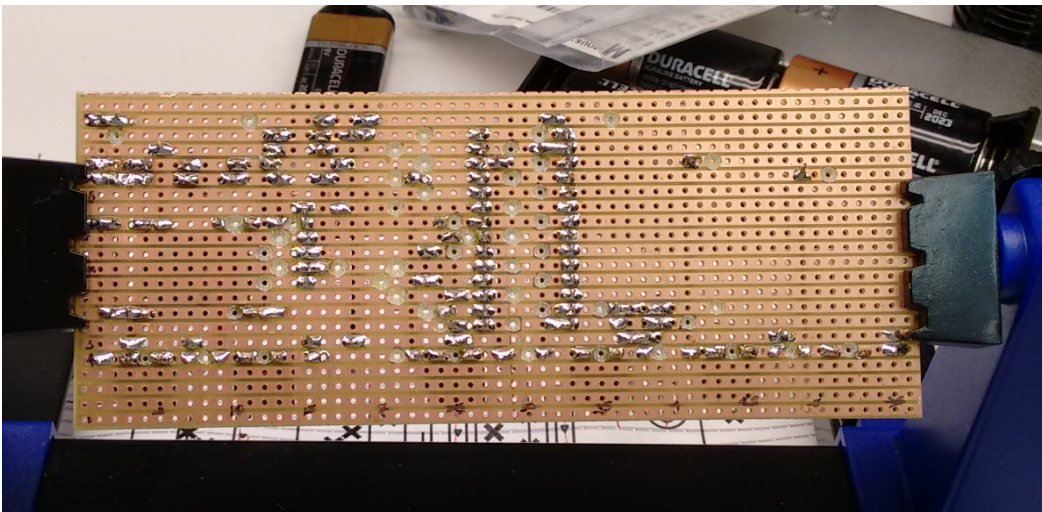
As with other boards, you want to start w/ the lowest profile components and work your way up. So I started with the jumper wires:



I then worked my way thru all the components:



here is the bottom. Not pretty, but not the worst solder job I've done:



All of the cheap components were soldered in. The more expensive stuff is socketed.

Test the Board

Next I test the board to verify voltages are where they should be, especially on MCU and other socketed chips.

Once I believe everything is in order, I insert the socketed chips and proceed with final testing.