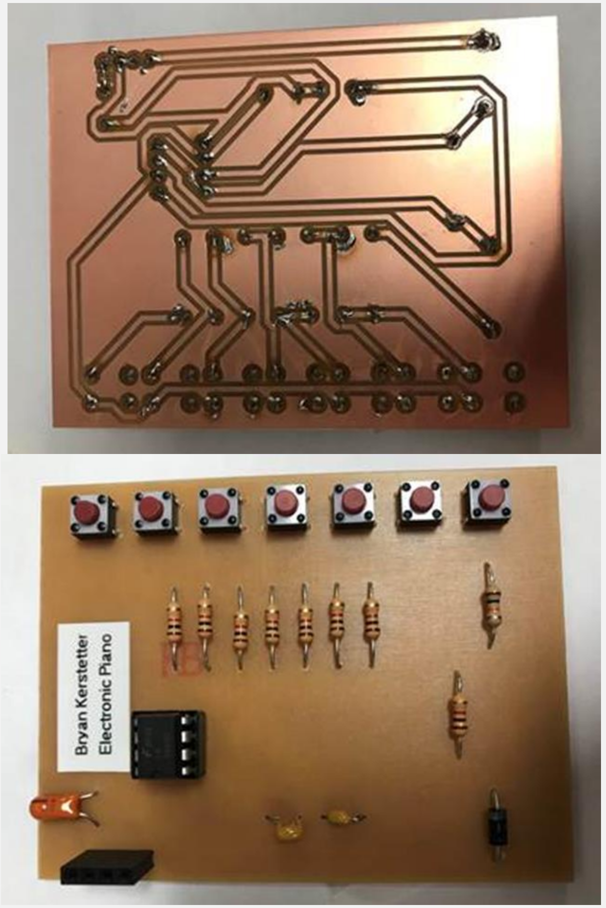
03-10-2020

AN INTRODUCTION TO PRINTED CIRCUIT BOARD DESIGN

Bryan Kerstetter

Dr. Grzegorz Chmaj

WHAT? WHY?



A milled Electronic Piano PCB

What?

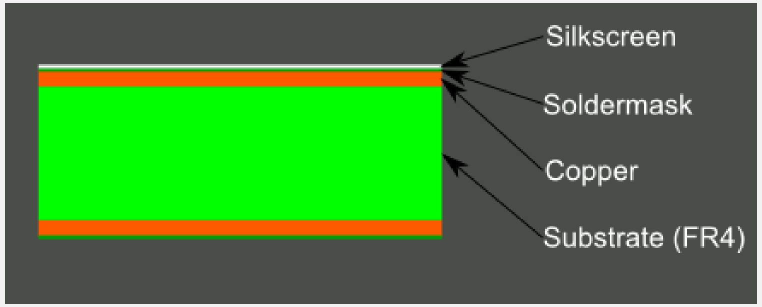
- A printed circuit board (PCB) is a board where components can be soldered to it such that after soldering the components are connected by copper traces.

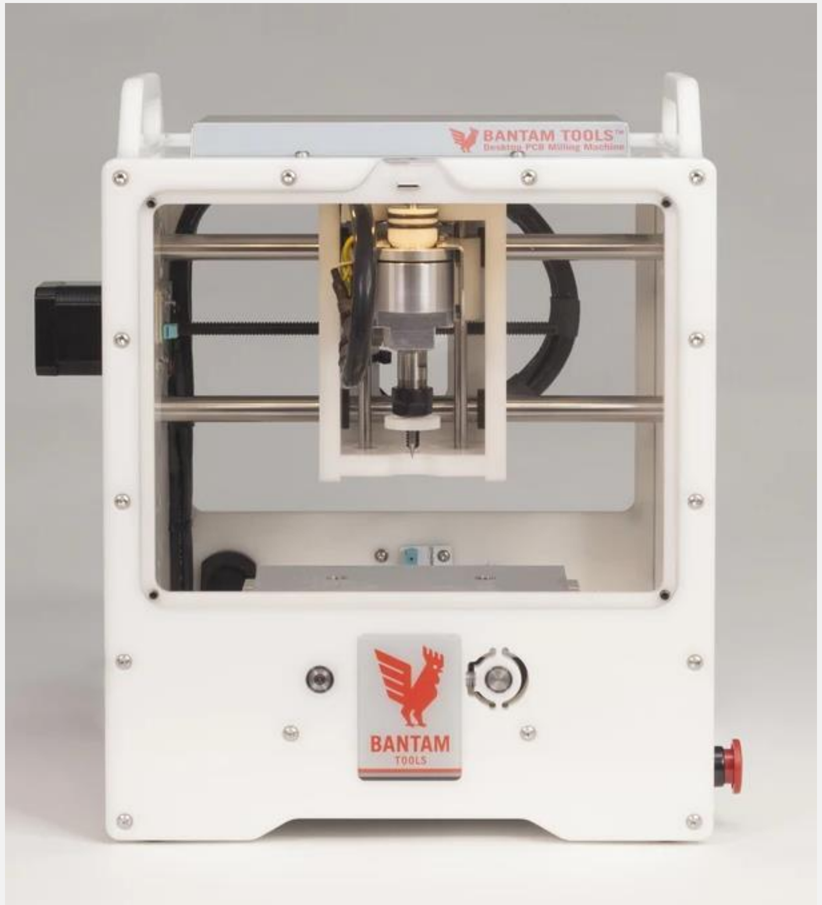
Why?

- A permanent solution for a finalized circuit
- Helps reduce noise
- Smaller size
- Easy to produce multiple copies

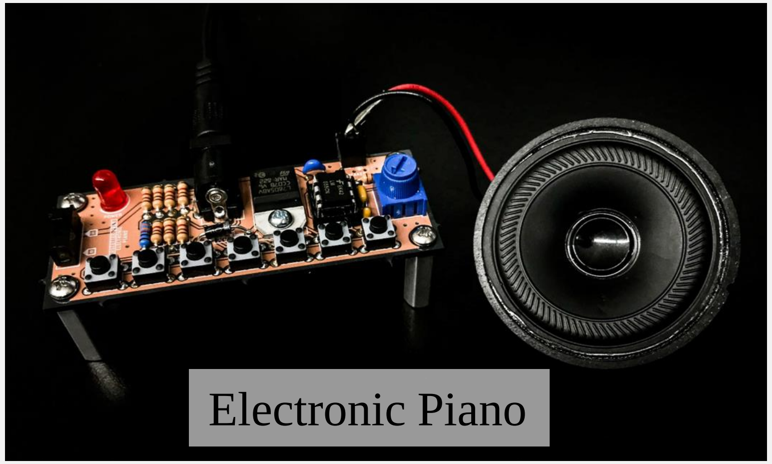
How?

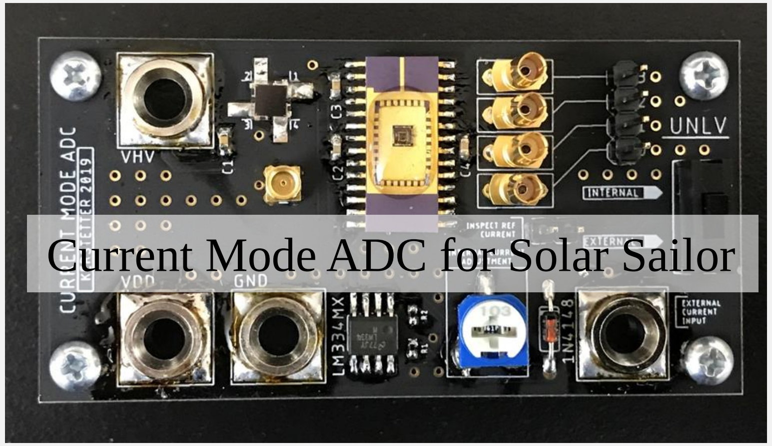
- Traces are milled on a copper plater (see milled PCB to the left)
- Standard fabricated PCBs also implement additional layers
    - **Soldermask:** A layer that is placed on top of the copper to prevent solder from adhering in those areas (in most PCB software this layer is a negative layer{in EAGLE: *tstop* and *bstop*}).
    - **Silkscreen:** A layer of ink that is printed on top of the soldermask. Used for printing values and instance names on the board.





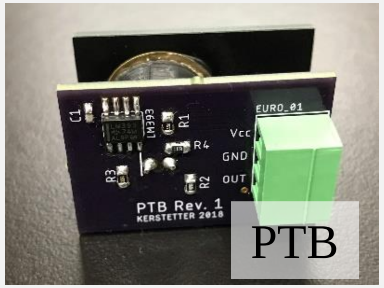
Bantam Tools Desktop PCB Milling Machine


Electronic Piano



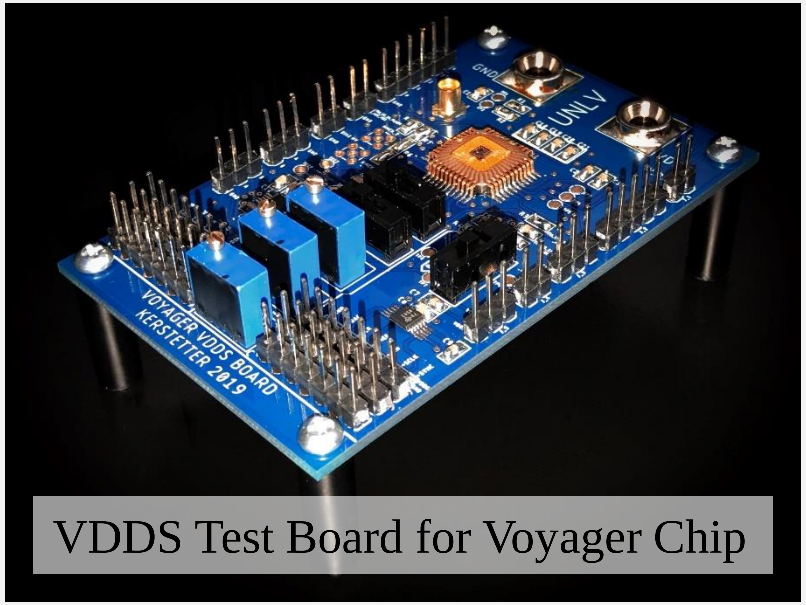
Current Mode ADC for Solar Sailor

LED Driver



PTB

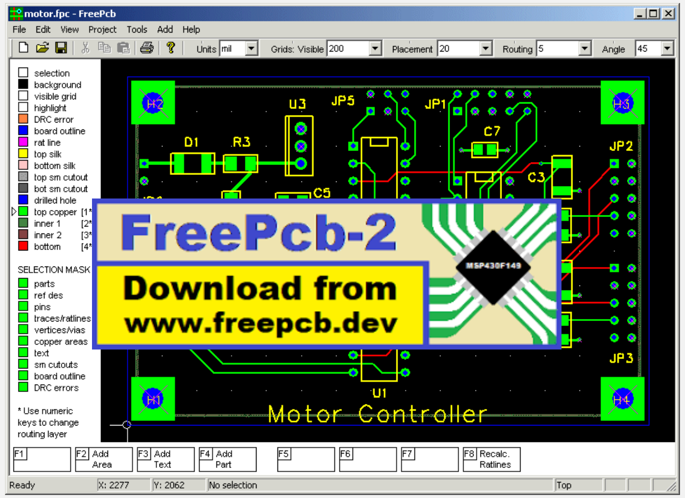
32x32 APD Camera System

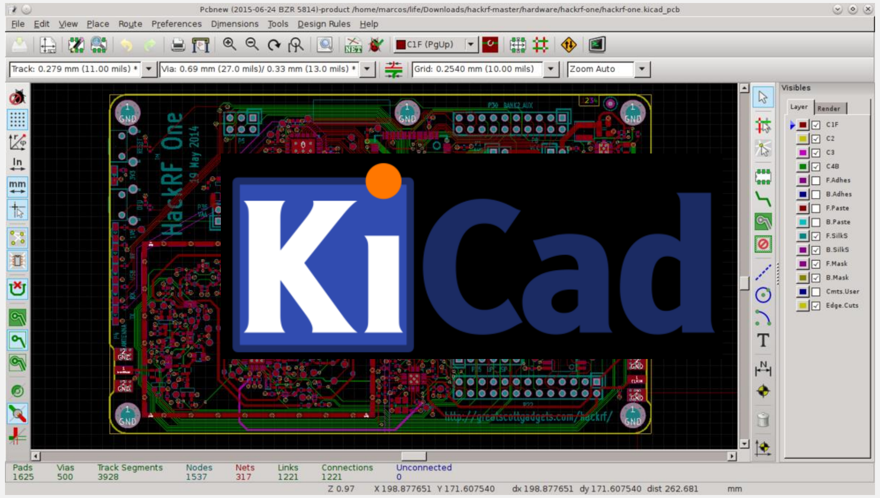


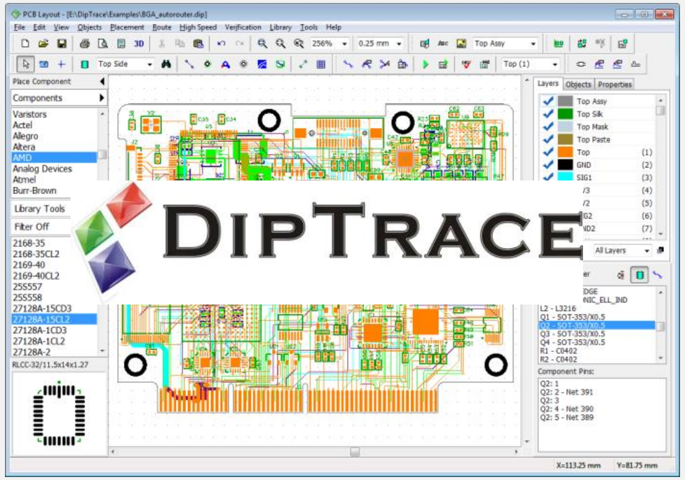
VDDS Test Board for Voyager Chip

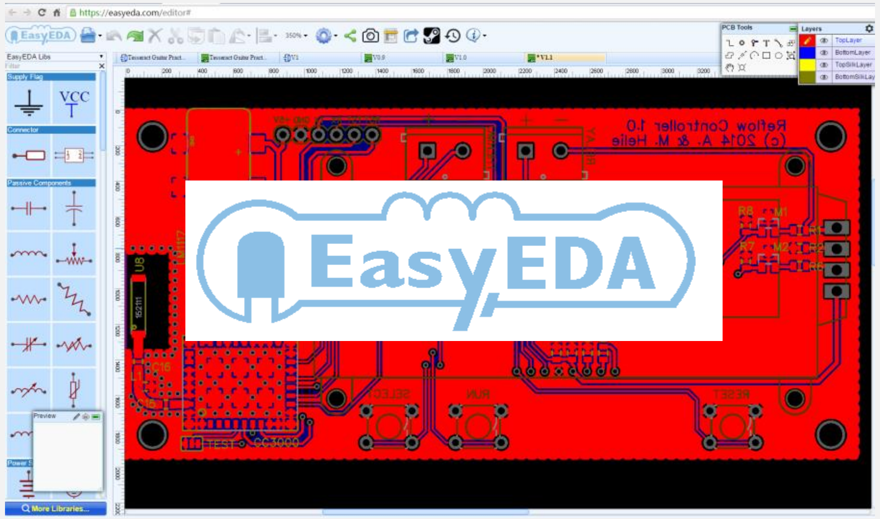
UNLV ECE AVR Development Board Rev. 2

WHAT SOFTWARE?



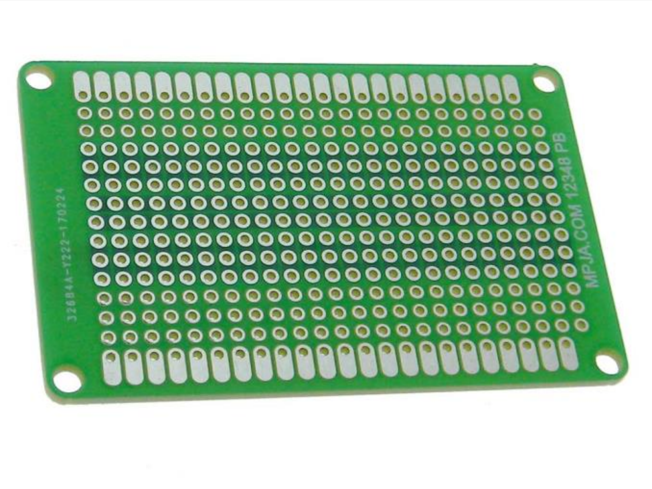






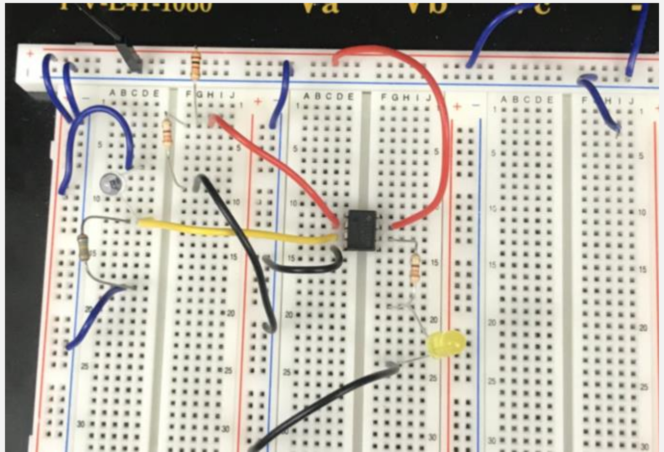
- Free Printed Circuit Board Design Software
  - FreePCB
  - DipTrace
  - KiCad
  - EasyEDA
- **HOWEVER, as a student you have access to AUTODESK EAGLE for free for 3 years**
  - This is by far the most feature rich PCB Design software you can use for free
  - **<u>I strongly recommend Eagle</u>**
  - Many companies make available their EAGLE PCBs
    - SparkFun
    - Arduino
  - Support
  - Ease of Use

PREREQUISITES

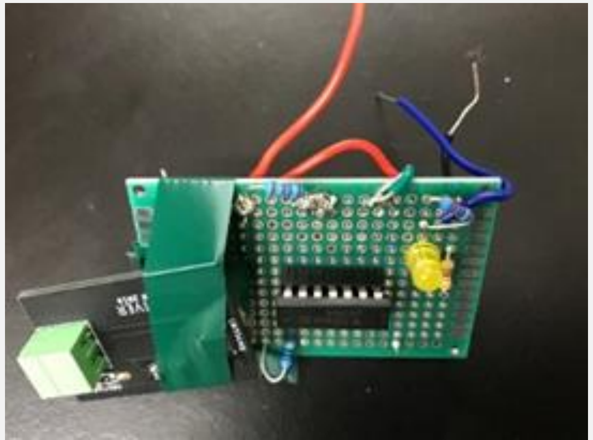


- **A complete working circuit must be completed**
  - Preferably, tested on a breadboard and confirmed to work
  - Sometimes not all portions of circuit can be tested on a breadboard, in which case test the portions that you can on the breadboard
  - Building your design on a proto board may be a good intermediary step
  - After all connections of your circuit is confirmed, create a schematic

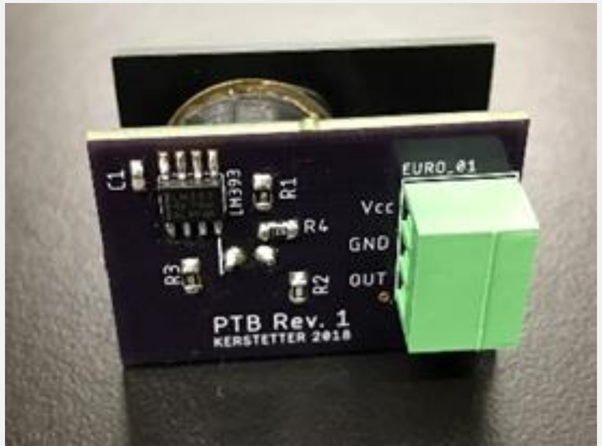
1. Bread Board



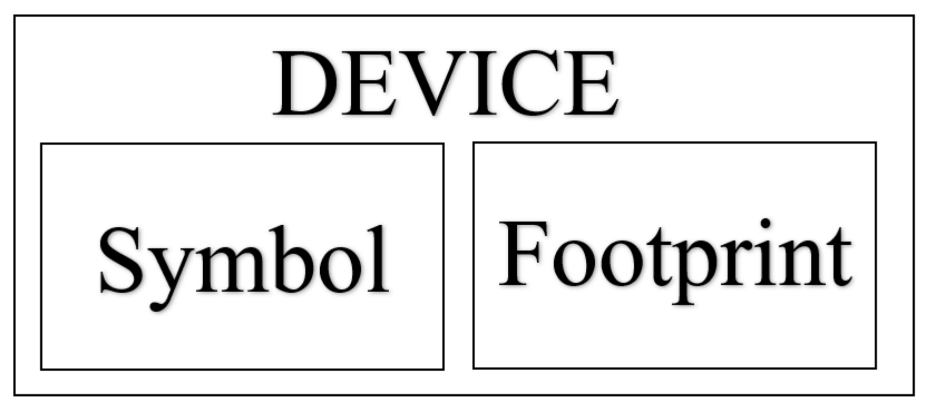
2. Proto Board



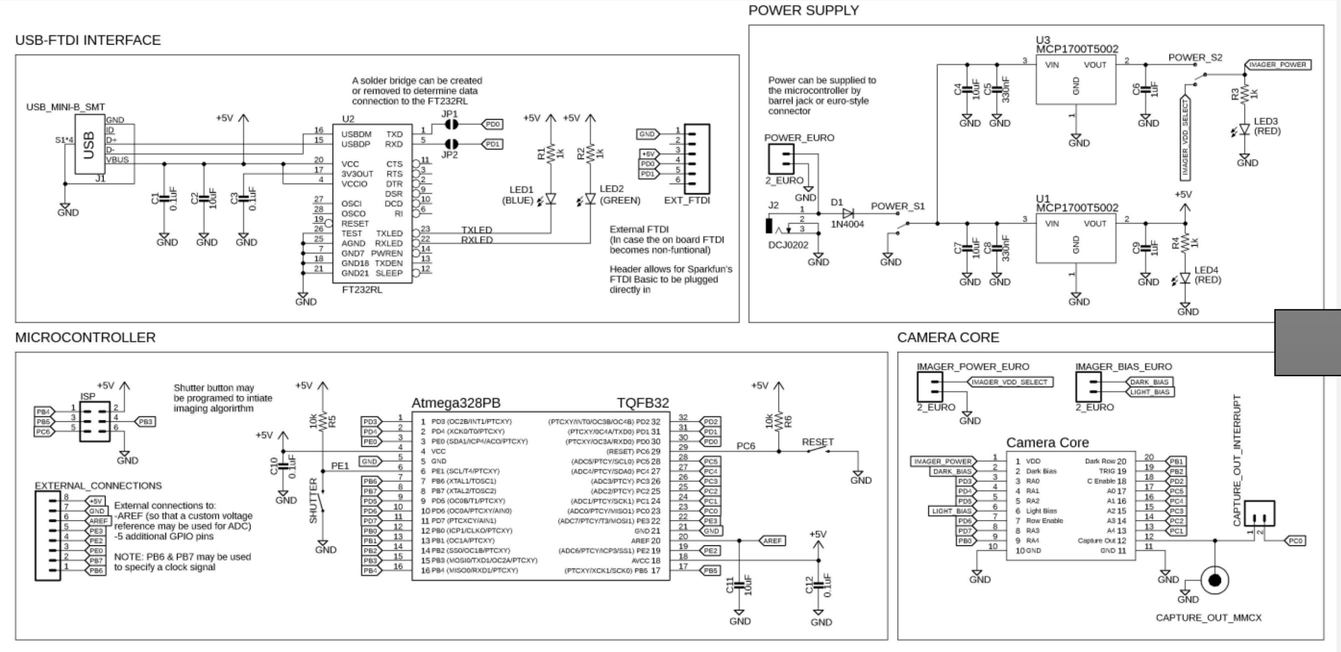
3. PCB

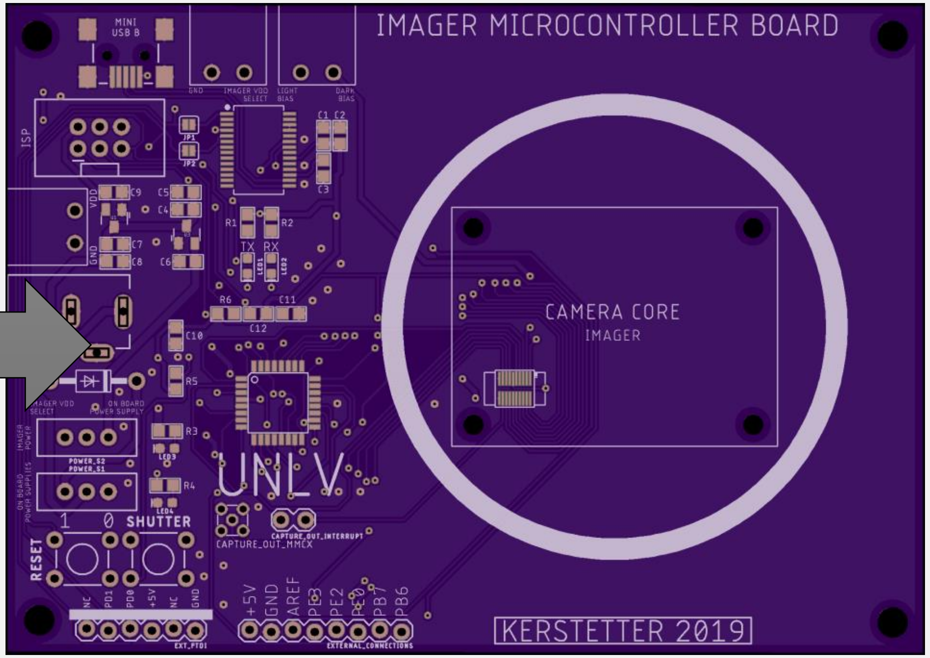


CREATING A SCHEMATIC

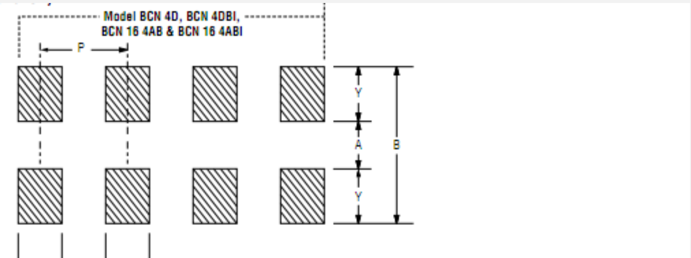


- If you have completed your design, then you know all the components that you need
  - Each component must have a respective device created for it
- Make your schematic clear to read
- Feel free to make notes on your schematic



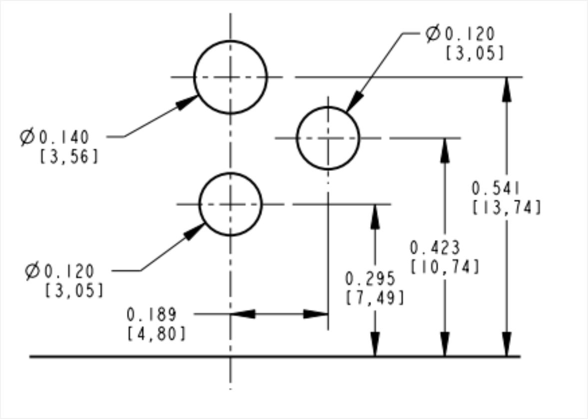


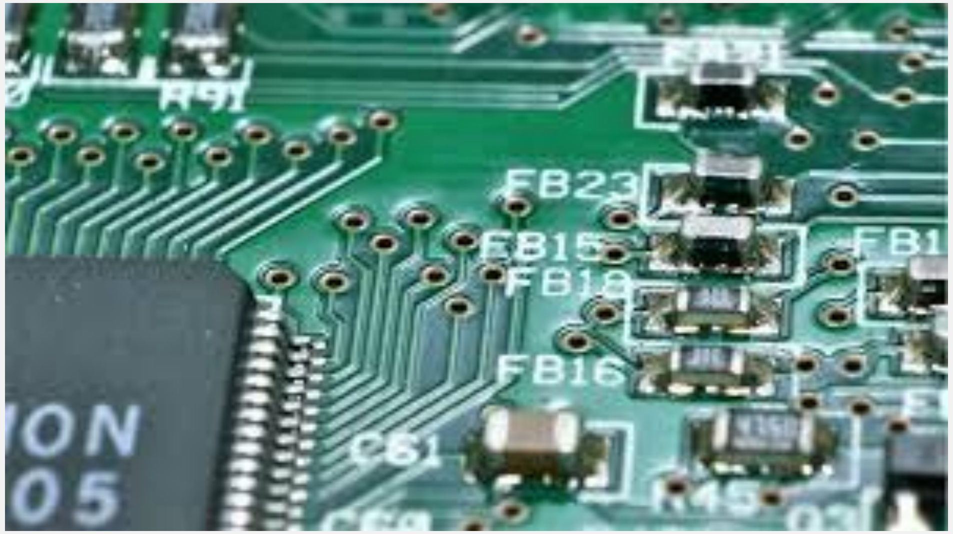
SURFACE MOUNT VS. THROUGH HOLE DEVICES

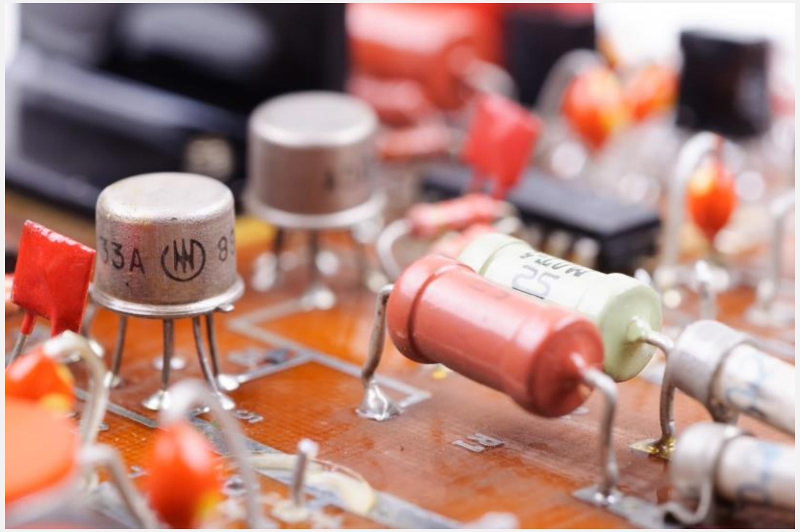


	P	Wave Solder Process					Re-Flow Solder Process				
		A	B	C	X	Y	A	B	C	X	Y
BCN 4D/4DBI	.050 / 1.27	.087 / 2.20	.169 / 4.30	.022 / 0.57	.028 / 0.70	.041 / 1.05	.087 / 2.20	.154 / 3.90	.022 / 0.57	.028 / 0.70	.034 / 0.85
BCN16 4A/AB/ABI	.032 / 0.80	.039 / 1.00	.118 / 3.00	.014 / 0.35	.018 / 0.45	.039 / 1.00	.039 / 1.00	.118 / 3.00	.014 / 0.35	.018 / 0.45	.039 / 1.00
BCN 31 8R/8S	.050 / 1.27	.084 / 2.10	.148 / 3.70	.014 / 0.35	.036 / 0.90	.032 / 0.80	.084 / 2.10	.148 / 3.70	.014 / 0.35	.036 / 0.90	.032 / 0.80

- Surface Mount Devices (SMD)
  - The device is soldered on one side of the board
  - Smaller, but harder to solder
    - Sometimes microscope is necessary to solder
    - Maybe consider stencil, solder paste, and oven
- Through Hole Devices
  - The device is attached by wires going through a hole on the PCB. Device attached to one side and soldered on the other.
  - Bigger, but easier to solder

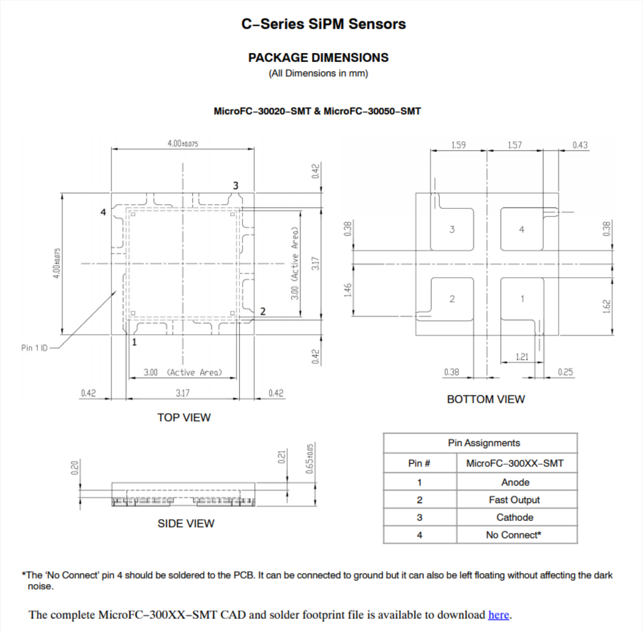


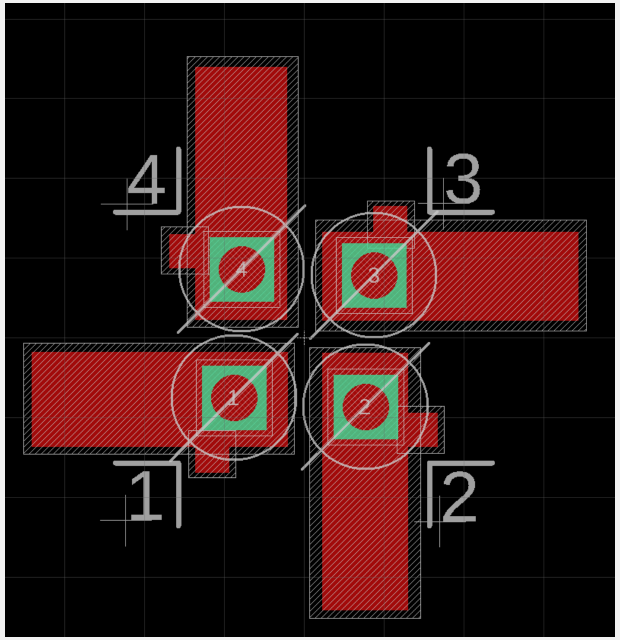


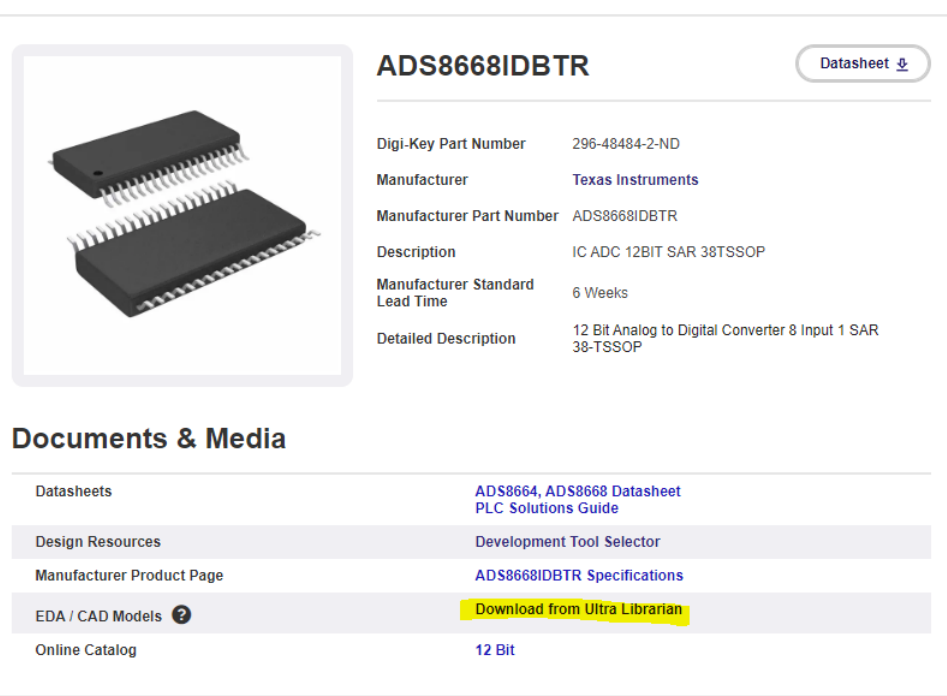


FOOTPRINTS

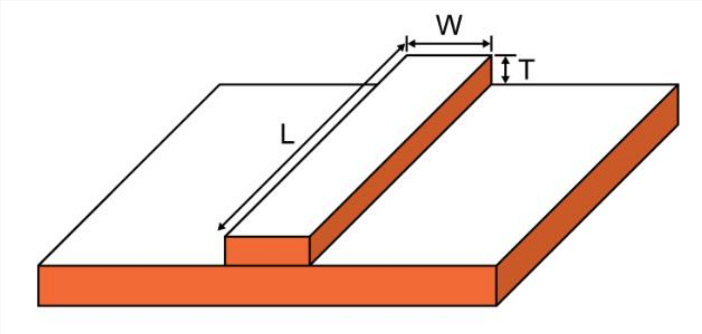
- Footprints are sometimes already available
  - Check DigiKey product page
- Be ready to make a footprint based upon drawings given in component datasheet







TRACES



- A PCB trace can be defined by the trace's width (W), length (L) and thickness (T)
- The trace will have resistance…

$$R = \rho \cdot \frac{L}{T \cdot W}[1 + \alpha \cdot (temp - 25)]$$

Where,

$$\rho = resistivity\ (copper)$$
$$L = length\ of\ trace$$
$$W = trace\ width$$
$$T = trace\ height$$
$$\alpha = temperature\ coefficeint\ of\ material\ (copper)$$

- As a PCB designer, you will only have control of L and W
- Therefore, if one assumes that

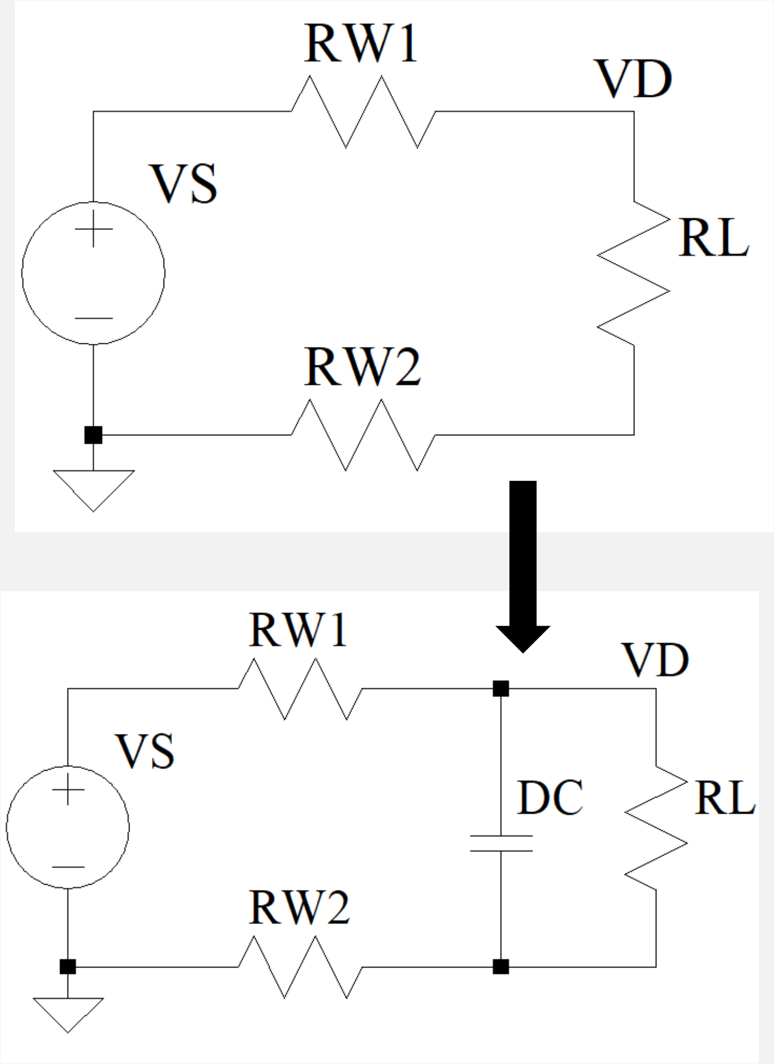
$$R_{\blacksquare} = \left[\frac{\rho}{T}[1 + \alpha \cdot (temp - 25)]\right]_{temp=20°C}$$

Then,

$$\boldsymbol{R = R_{\blacksquare} \cdot \frac{L}{W}}$$

SOURCE: https://www.allaboutcircuits.com/tools/trace-resistance-calculator/

TRACES CONTINUED I



A voltage supply (VS) delivers power to a microcontroller (RL), where supply is connected to the microcontroller by two traces.

$$RW1 = RW2 = R_{\blacksquare} \cdot \frac{L}{W}$$

$$VS - 2\left(R_{\blacksquare} \cdot \frac{L}{W}\right) \cdot I - RL \cdot I = 0$$

$$\frac{VS}{2\left(R_{\blacksquare} \cdot \frac{L}{W}\right) + RL} = I$$

$$VD = \frac{VS \cdot RL}{2\left(R_{\blacksquare} \cdot \frac{L}{W}\right) + RL}$$

Conclusion

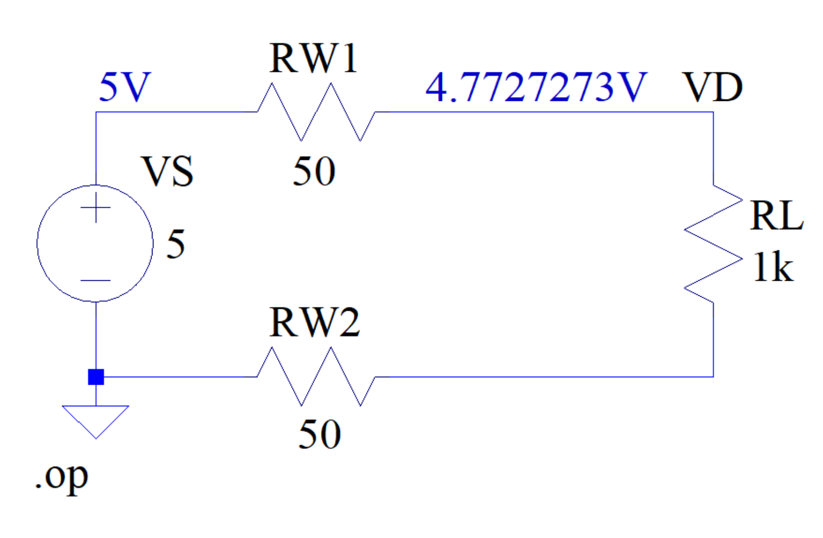
$$VD \propto W$$

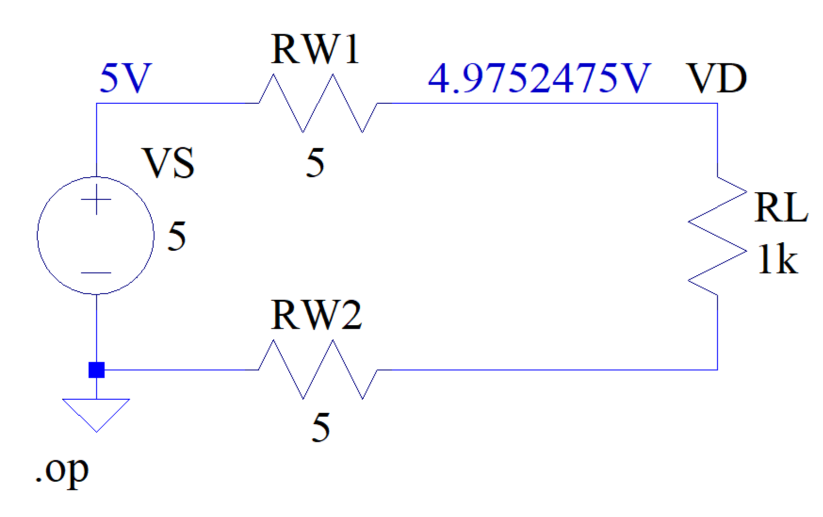
$$VD \propto \frac{1}{L}$$

Therefore, you want to make traces as wide as possible (especially power traces).

Furthermore, decoupling capacitors should be added as well. Decoupling capacitors, ensure that the load has steady current despite possible variances and noise in the source.

TRACES CONTINUED II





Assume that the PCB copper sheet resistance is $1\Omega/\blacksquare$. And both traces are 100mm (~3937 mil) in length and 2mm (~79 mil) in width. What is the voltage across the microcontroller if the microcontroller's resistance is 1kΩ?

Therefore, according to…

$$RW1 = RW2 = R_{\blacksquare} \cdot \frac{L}{W} = 1 \cdot \frac{100\ mm}{2\ mm} = \mathbf{50\Omega}$$

Voltage across load: 4.77V

Let's make the trace width 20mm…

$$RW1 = RW2 = R_{\blacksquare} \cdot \frac{L}{W} = 1 \cdot \frac{10\ mm}{20\ mm} = \mathbf{5\Omega}$$

Voltage across load: 4.98V

CONCLUSION

AGAIN, make power traces WIDE and all traces, for that matter, WIDE

TRACES CONTINUED III

Current (A)	Trace Width (mil)
1	10
2	30
3	50
4	80
5	110
6	150
7	180
8	220
9	260
10	300

ADVANCED CIRCUITS | 4PCB

4PCB Calculator

Inputs:

Current	10	Amps
Thickness	2	oz/ft^2

Optional Inputs:

Temperature Rise	10	Deg C
Ambient Temperature	25	Deg C
Trace Length	1	inch

Results for Internal Layers:

Required Trace Width	368	mil
Resistance	0.000685	Ohms
Voltage Drop	0.00685	Volts
Power Loss	0.0685	Watts

Results for External Layers in Air:

Required Trace Width	142	mil
Resistance	0.00178	Ohms
Voltage Drop	0.0178	Volts
Power Loss	0.178	Watts

- There are tables and calculators that can help you know how to size your traces

ADDITIONALLY,

There is a difference between a mil and a mm.

MIL (otherwise known as *THOU*)

Measurement in Imperial System

1/1000th of an inch

MM

Measurement in the metric system

1/1000th of a meter

Recommended Trace Sizes

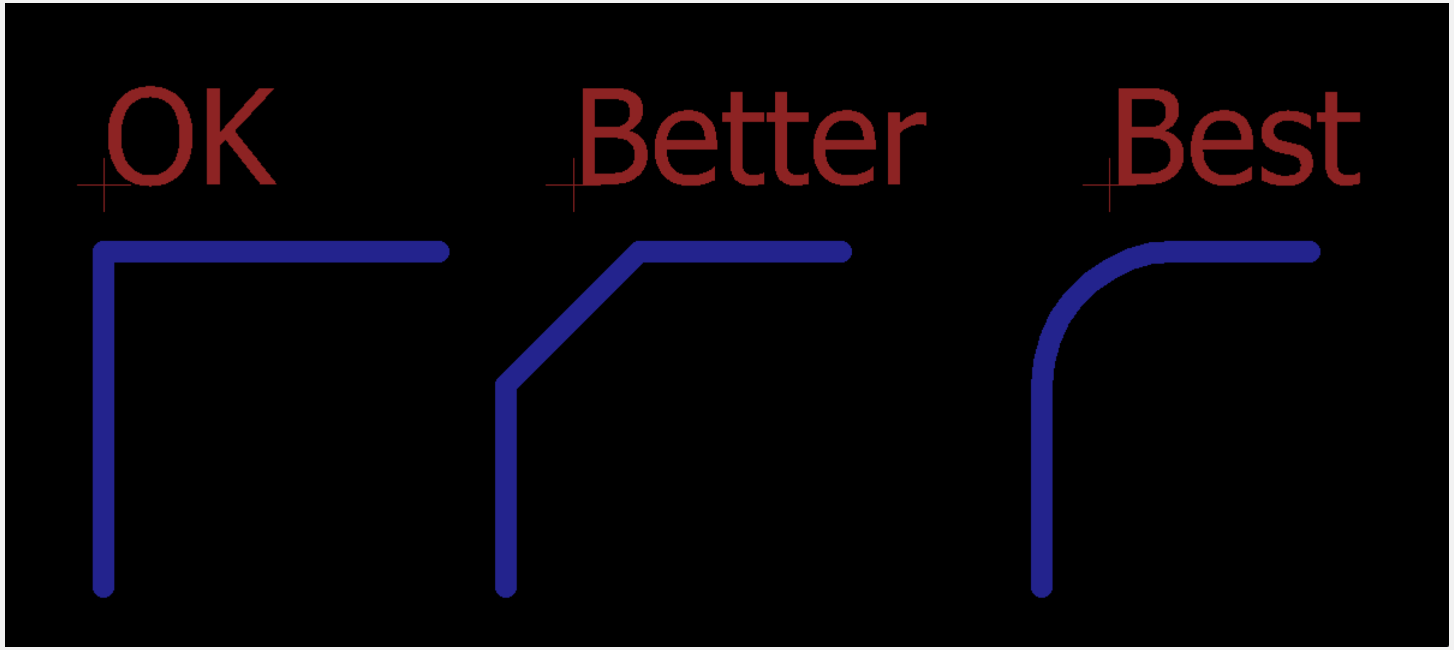
50 mil	Power Traces
25 mil	Signal Traces
10-15 mil	Between ICs and Pads

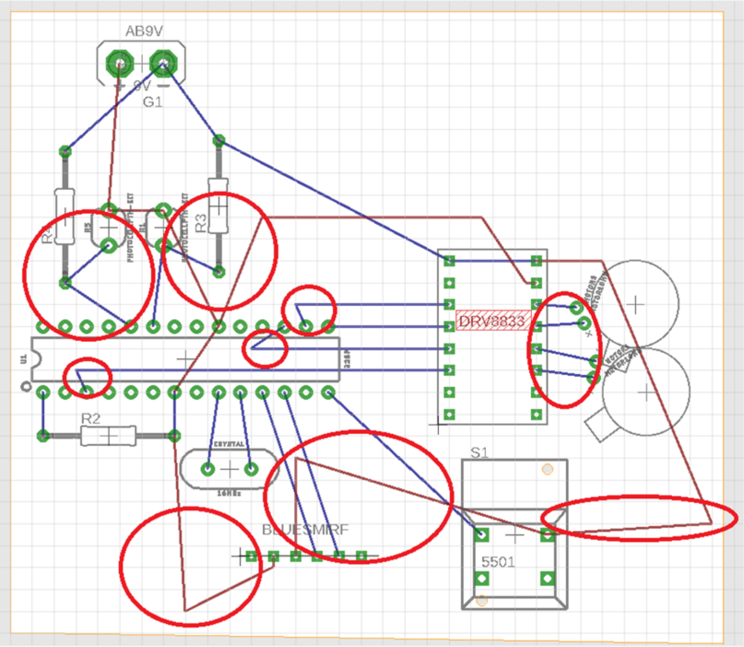
SOURCE: https://resources.altium.com/pcb-design-blog/pcb-trace-width-vs-current-table-for-high-voltage-design

TRACES CONTINUED IV

$\geq 90°$

- **No trace angles should be acute (less than 90°)**
- **Trace angles should be preferably greater than 90°**

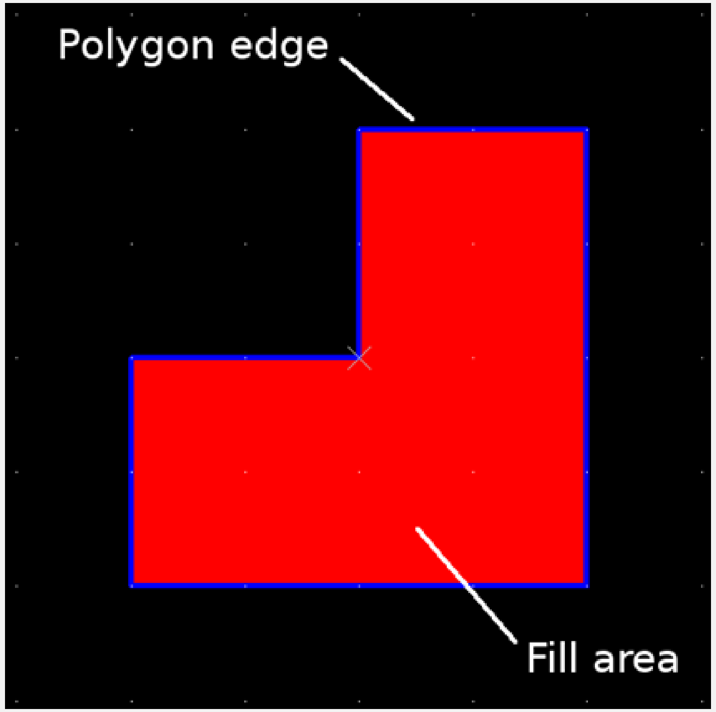


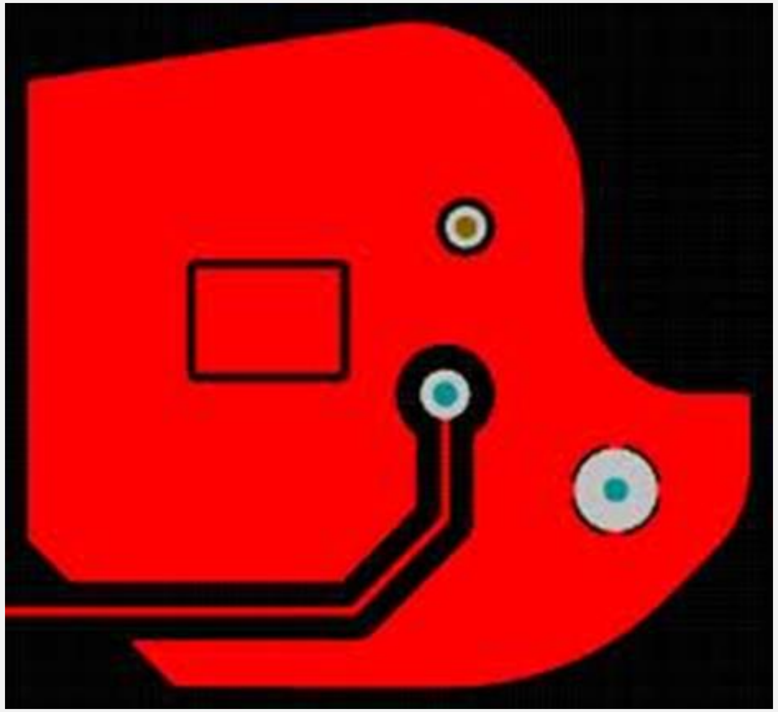


SOURCE: https://www.autodesk.com/products/eagle/blog/top-10-pcb-routing-tips-beginners/

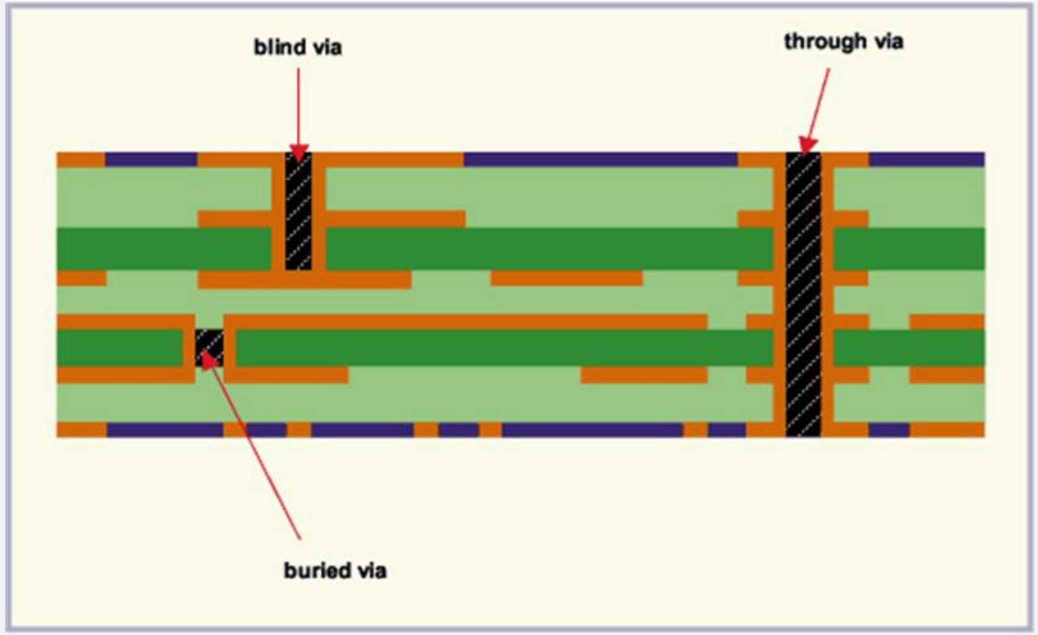
POLYGONS

- **Polygon automatically fills in a desired area with copper, which “flows” around other pads and tracks**
  - Useful for laying down ground planes
  - Make sure you place polygons after you placed all of tacks and pads





VIAS



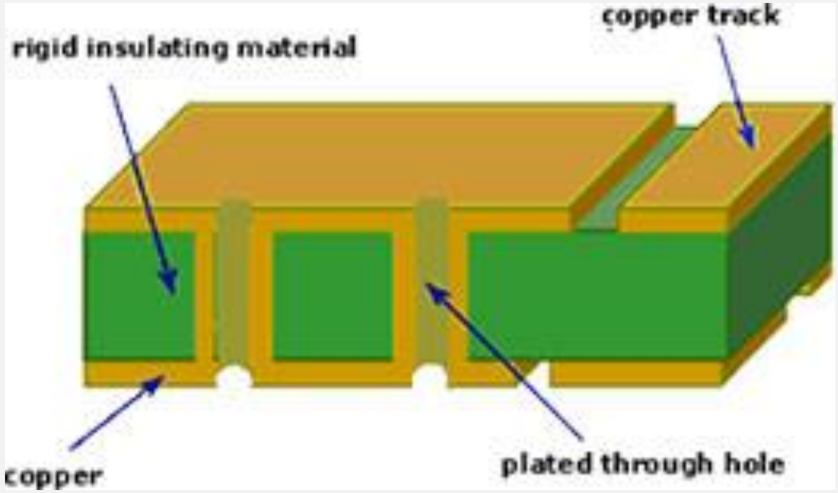
6 Layer PCB

2 Layers

- Vias connect the tracks from one side of the board to another through a hole in the board
  - Usually made with electrically plated holes – Plated Through Holes (PTH)

More Than 2 Layers

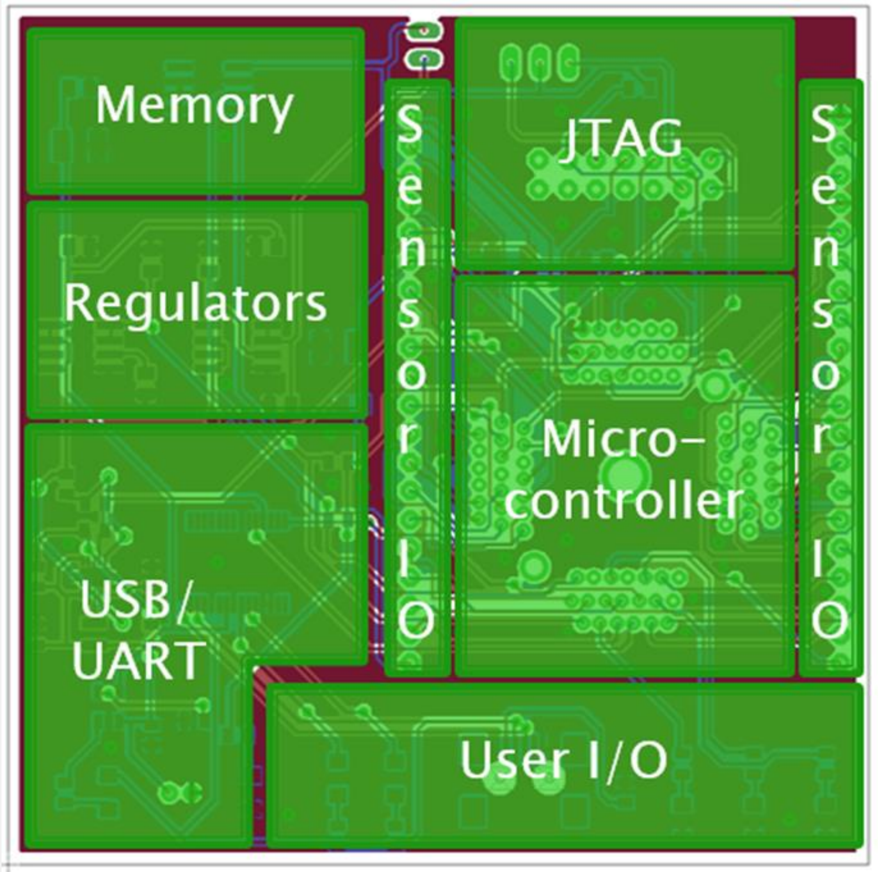
- Allow electrical connection between different layers of the board



2 Layer PCB

☆ Pins of through-hole components can be used a via!

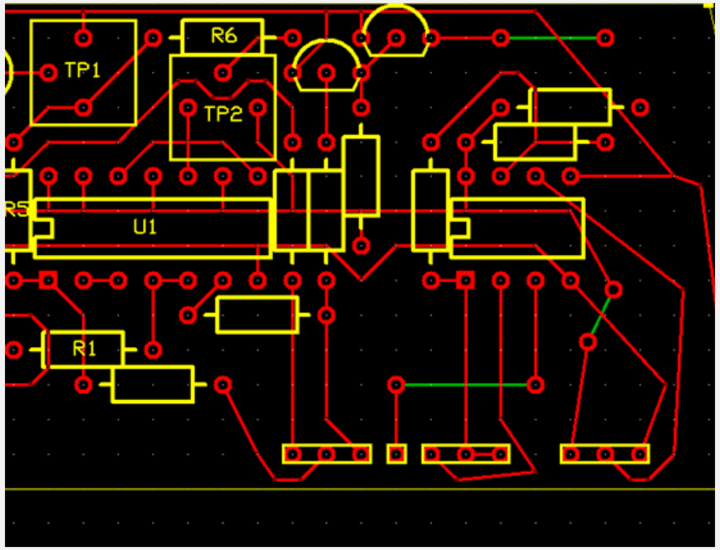
PCB FLOORPLAN

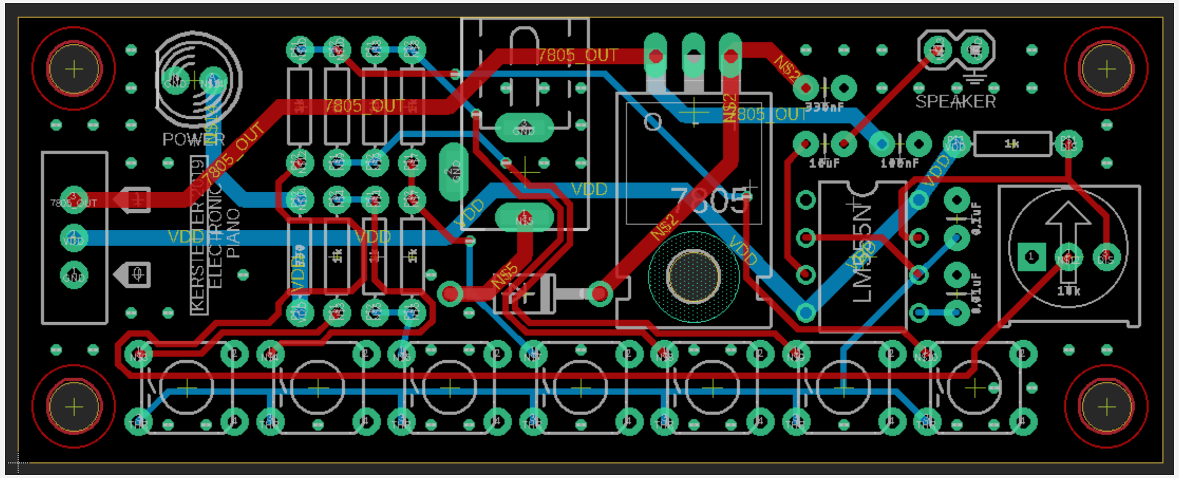


Sometimes it is a good idea to make a floorplan (according to function) of a PCB before components are placed.

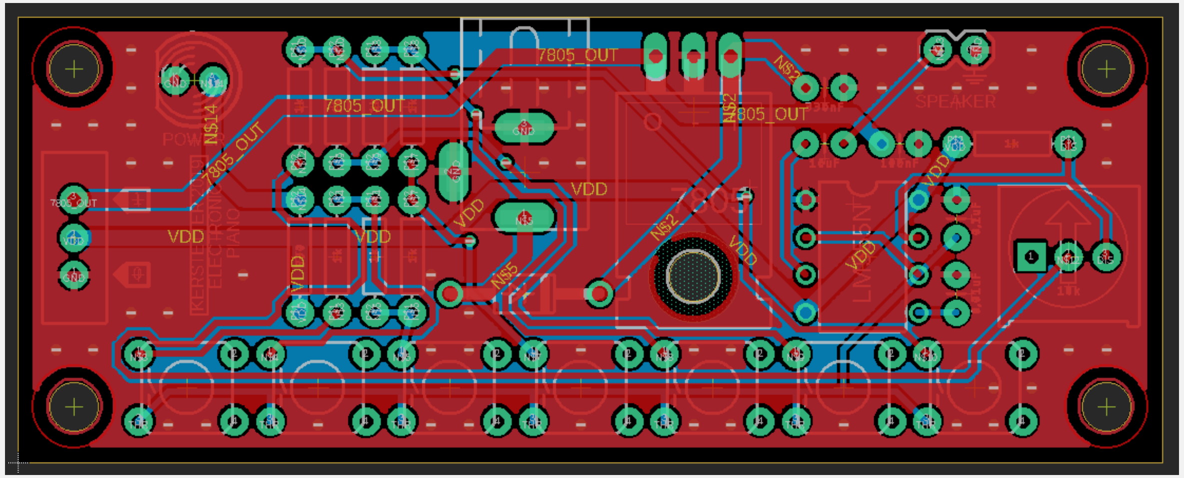
This is especially useful for larger, more complex designs.

GOOD VS BAD LAYOUT



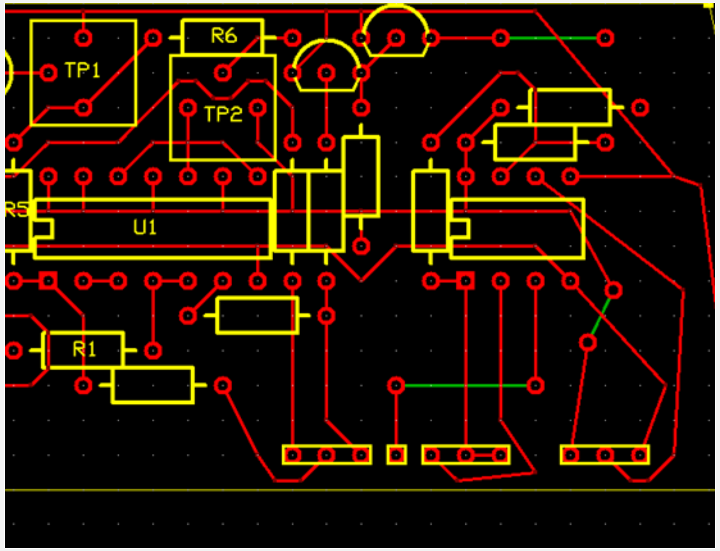


Power and ground plans implemented:



GOOD VS BAD LAYOUT

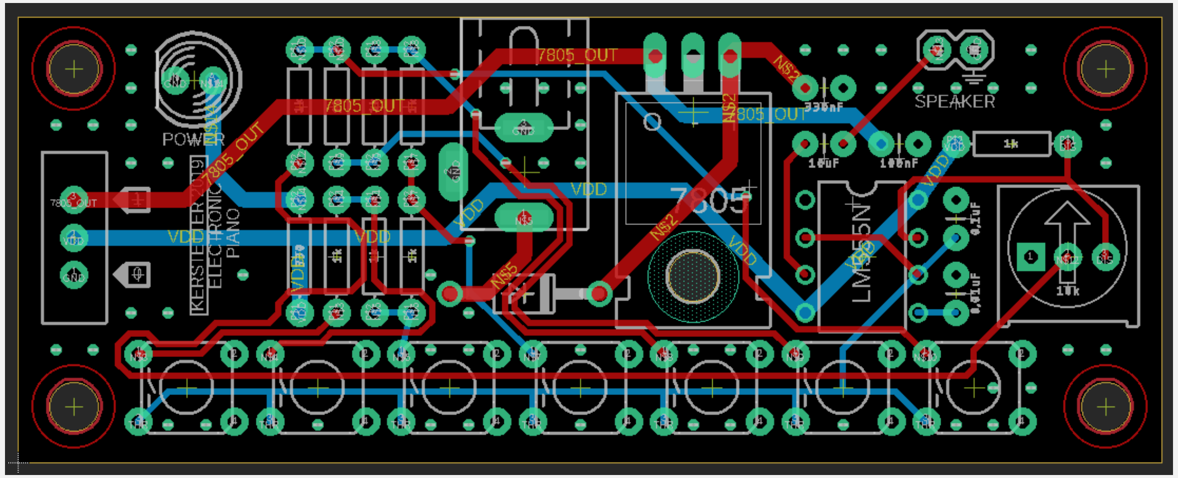
BAD LAYOUT



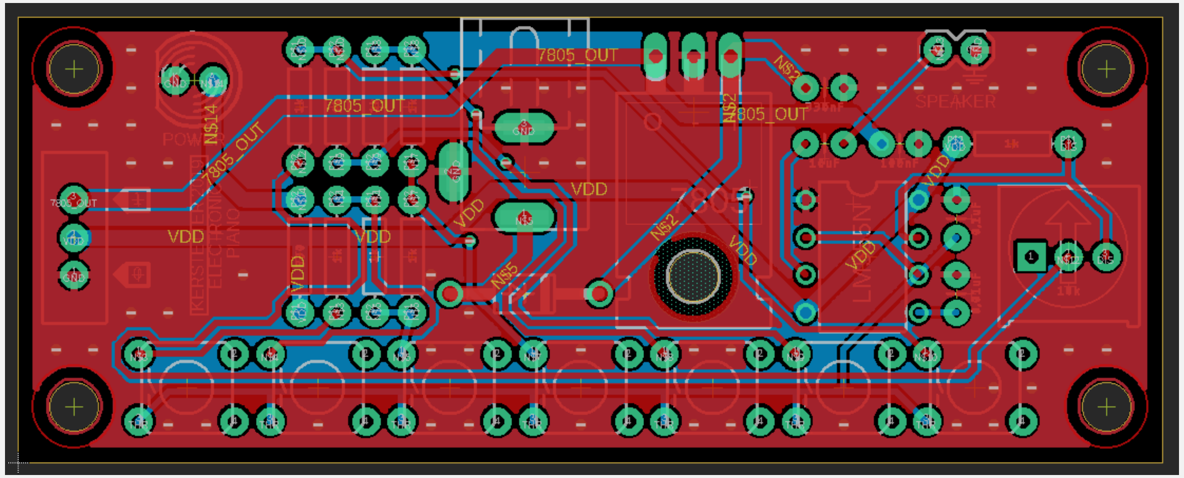
ISSUES:

- Traces are way too small (especially the power traces)
  - **If the circuit draws too much current, and the trace is to thin, the trace can burn up!**
- Traces have acute angles
- Inefficient use of space
- Poor placement of components

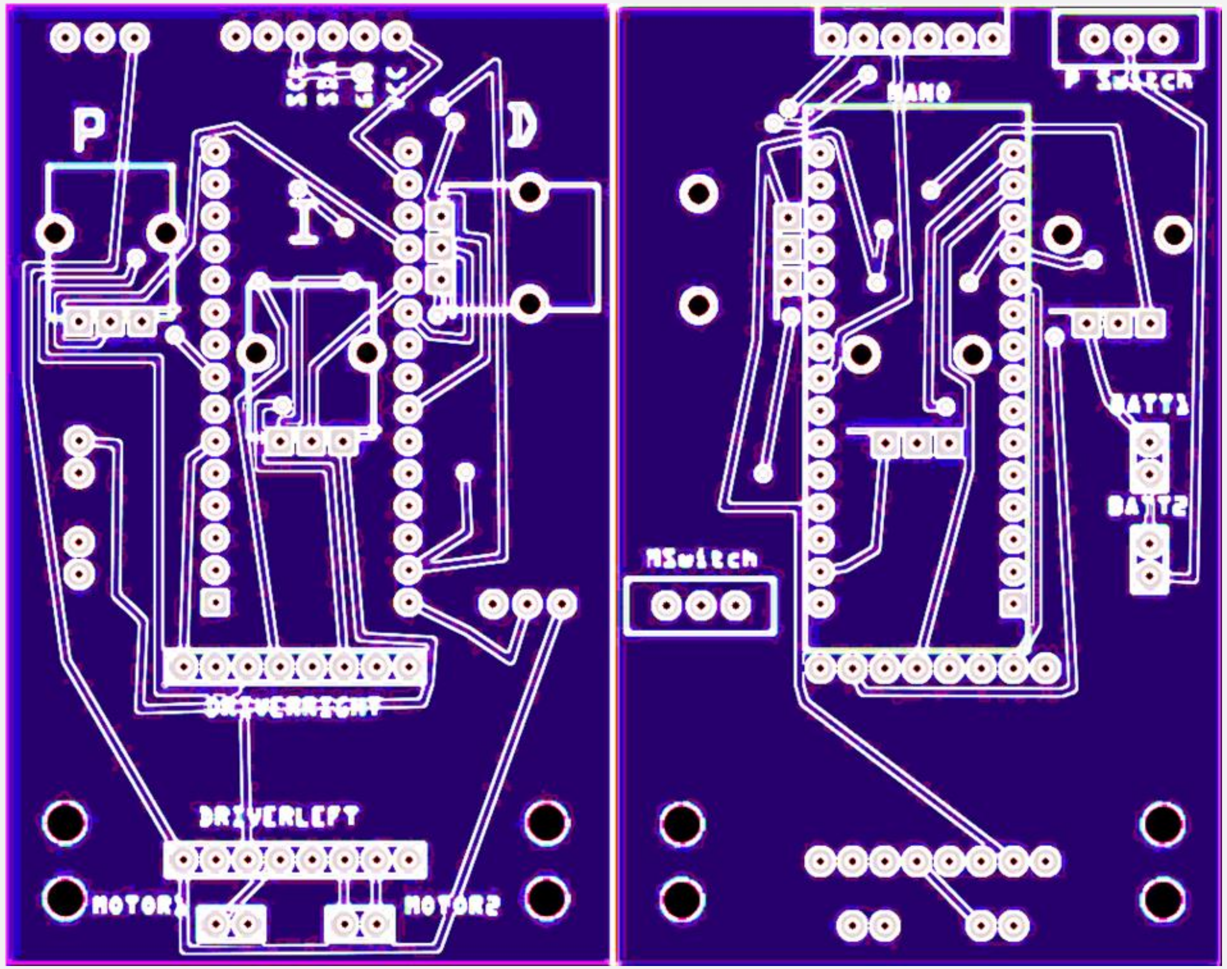
GOOD LAYOUT



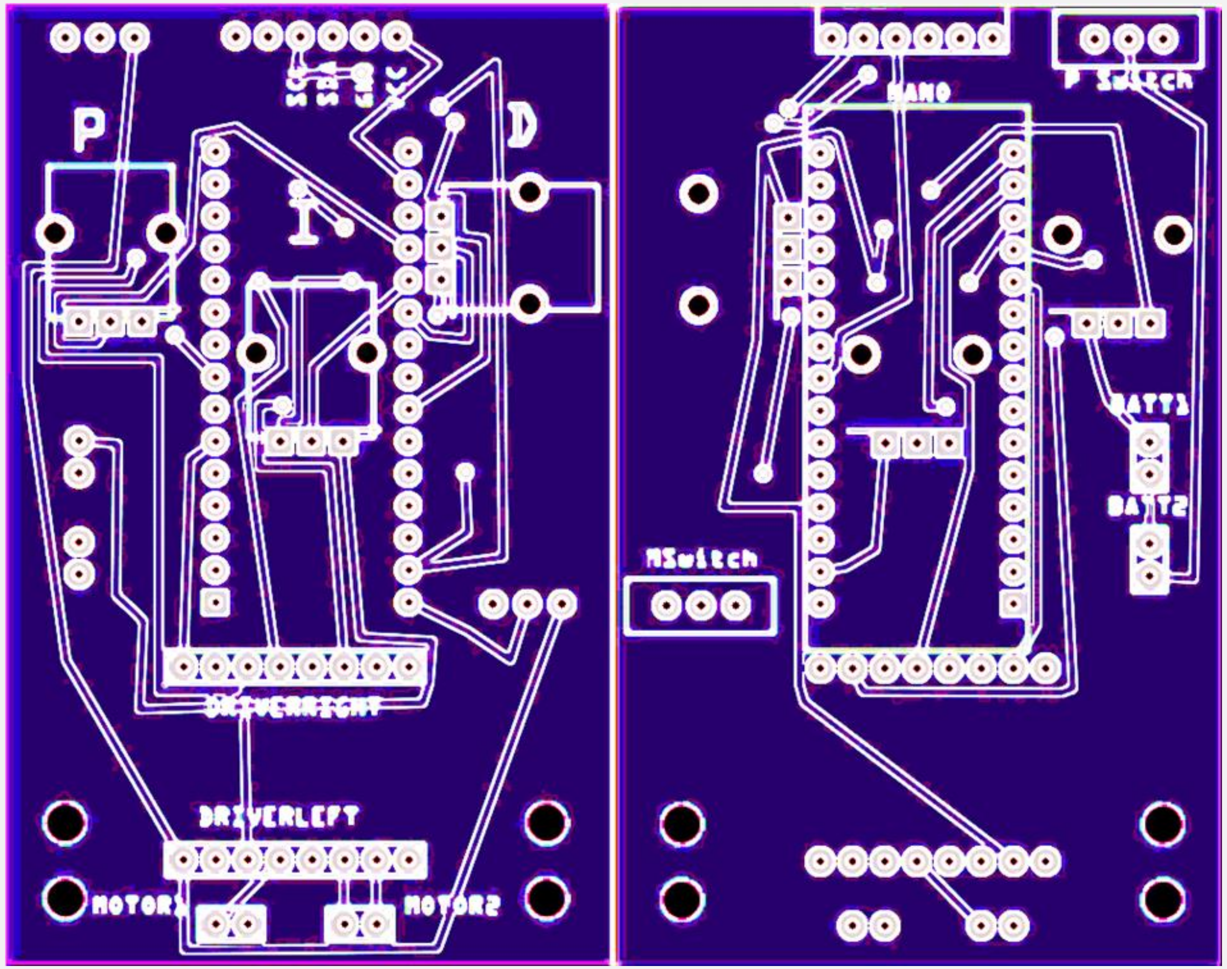
Power and ground planes implemented:



IS THIS LAYOUT…
GOOD? BAD?



IS THIS LAYOUT…
GOOD? BAD?

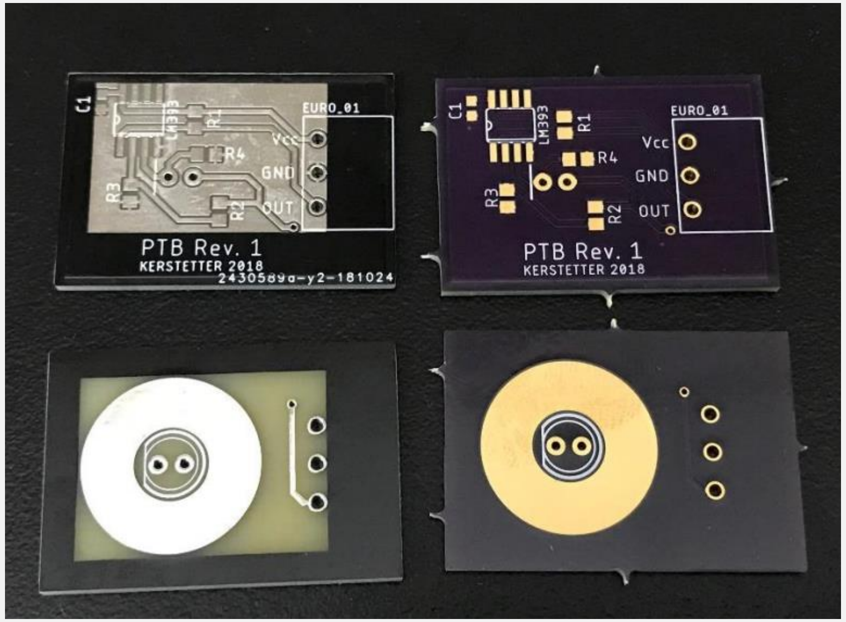


SAME ISSUES AS BEFORE:

- Traces are way too small
- Traces have acute angles
- Inefficient use of space
- Poor placement of components
- Board is much bigger than what it should be
    - **Smaller Board = Cheaper !**

LAST COMMENTS

- Consider the heating of your components
  - Are they necessary? Heat sinks?
- Avoid tracks with an “end” – as it becomes an antenna
- Make sure tracks go to the exact center of pads and components
- Consider having:
  - Two ground planes
    - Make sure to place periodic vias to connected the two ground planes (improves heat dissipation and ensures good ground connection)
  - A ground plane and a power plane
    - Helps when routing complex designs
- Remember, PCB price is determined by the size of the board
  - **MAKE YOUR BOARD AS SMALL AS POSSIBLE**
- Consider the PCB fabrication company
  - Does the company produce good quality PCBs?
  - I have had good experience with Osh Park



JLCPCB

- Cheap
- Poor Quality

OSH Park
Perfect Prompt Purple PCBs

- Reasonably Priced
- Great Quality

EAGLE DEMONSTRATION