

Home Prototyping

Producing Printed Circuit Boards at home

Doug Theriault - NO1D

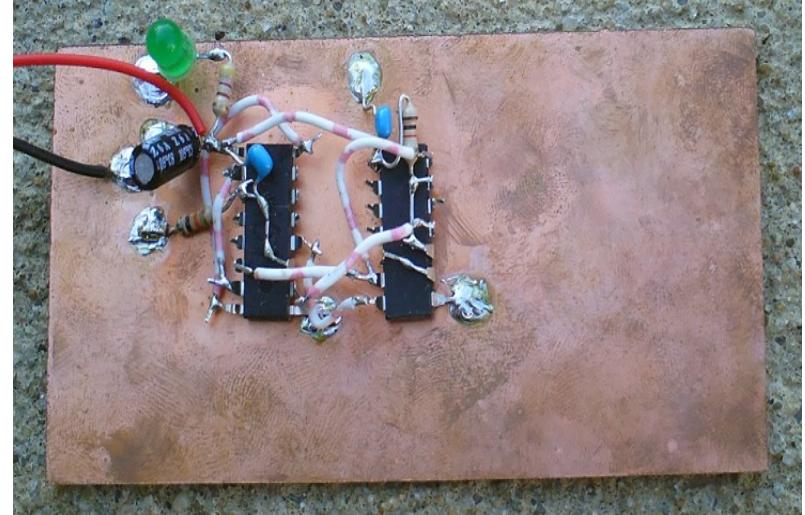
Agenda

- Present steps for the fabrication of PCB's at home using the Toner Transfer method.
- Schematic Capture, Board Routing example
- How to prepare and etch using HCL and H2O2
- Some Useful tools and equipment; pictures of my workspaces
- Safety tips, Helpful Hints, Gotchas
- Movie clip showing the process; (7m)
- Brief discussion on steps to make 2 sided PCB's
- A Few reference links/URL's
- Example circuit; A YARC Ham ID Timer
 - (12) finished boards for you to experiment with
 - (12) blank boards for those wanting to build from scratch

Typical Construction Methods

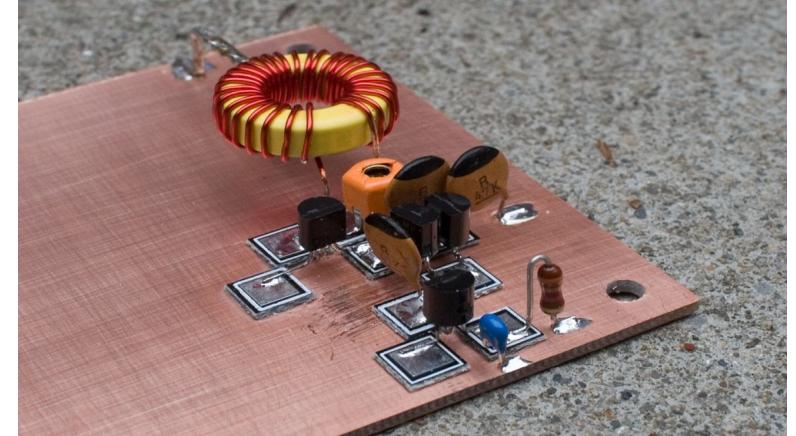
Ugly / Dead Bug

- Single Ground Plane
- Fast/Easy

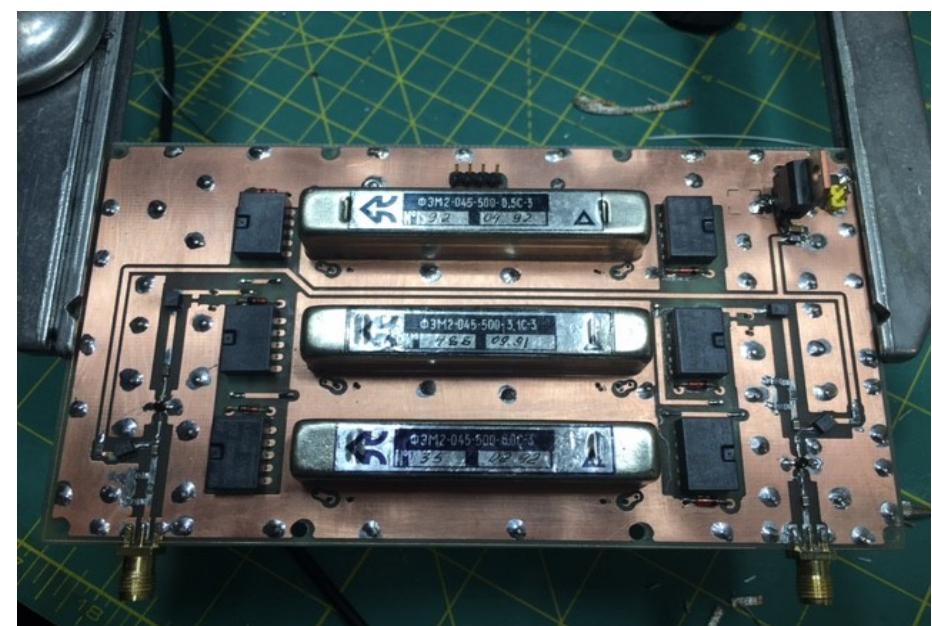
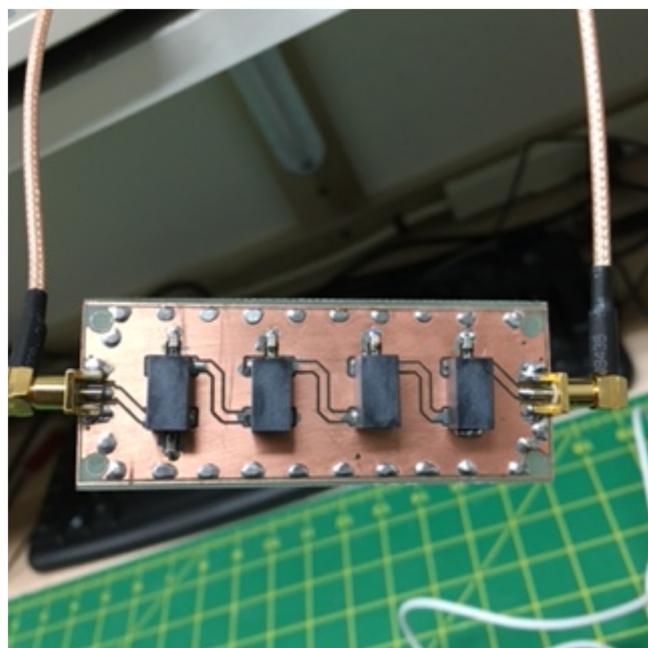
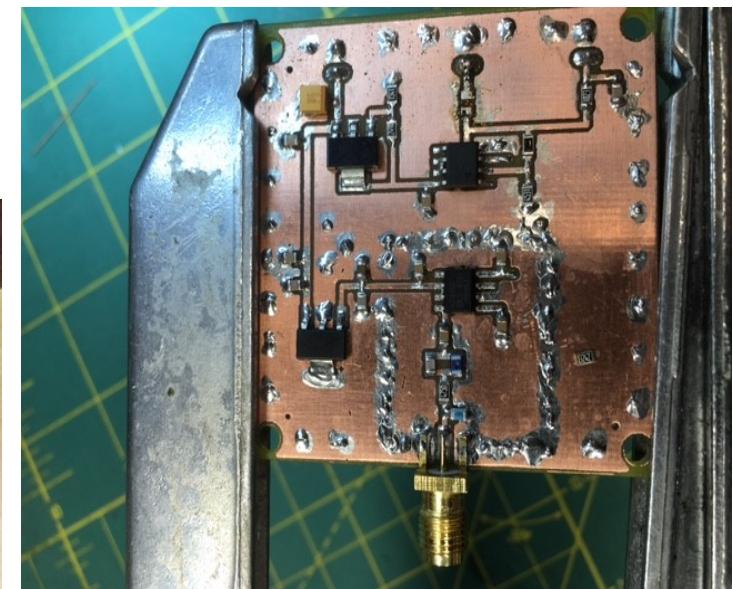
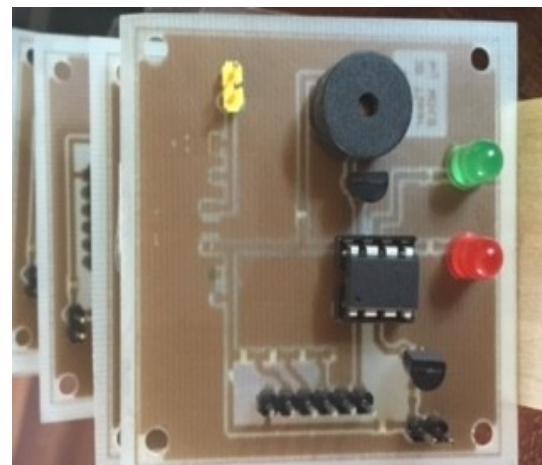
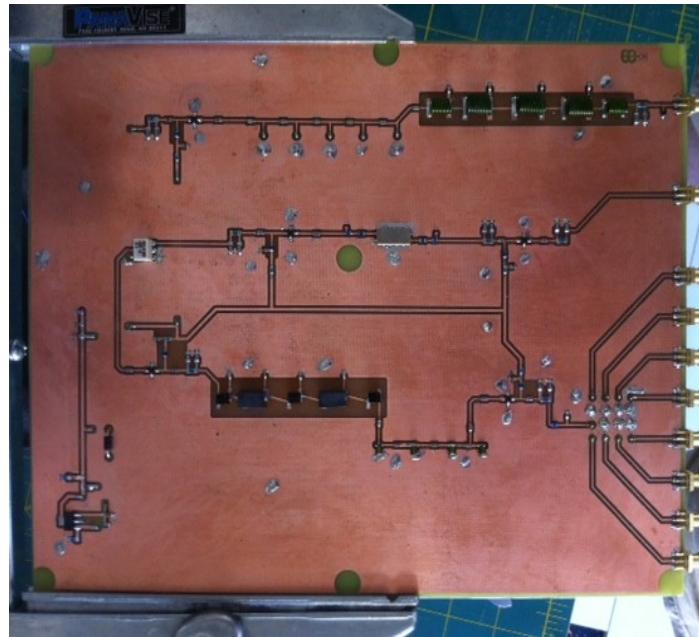


Manhattan Style

- Single Ground Plane
- Uses square/round isolation pads
- Can cut/ground pads onto ground plane w/ hacksaw or Dremel tool
- Precut pads
 - (W1REX @ qrpme.com)
- Can make very nice boards



Or Printed Circuit



Why Bother

- Further the art of home brewing
 - Satisfaction of building your own gear
 - Great learning experience
 - Improve look/feel/maintainability of home brew gear
- Specific to PCB's; Prove a designs before committing to a board house;
Tindie, OshPark
- Part of our Amateur Code/Spirit:
 - Considerate, Loyal, **Progressive**, Friendly, Balanced, Patriotic
 - Learn and follow in footsteps of some great Hams: W1FB, W7ZOI, KK7B and many many others !
- And... It can be Fun and a hit with the neighbors



Items Needed

- Schematic Capture Application
 - Eagle PCB or KiCad (others)
- Image Editing Software (optional)
- Laser Printer
- Paper – Dextrose Coated PCBFX
 - (optional: Toner Protection Dry coating also by PCBFX)
- Laminate Machine
- Various sized Pyrex Pan(s)
- Sharp Knife
- Straight Edge
- Blue Painters Tape
- Shelf Contact Paper
- Latex/Nytrol Gloves
- Paper Towels
- ISO-Propal Alcohol
- Green Scotch Bright Pad(s)
- Water
- HCL – Muriatic Home Depot
- Hydrogen Peroxide
 - 30% to 40% clear
- Measuring cup (not from kitchen!)
- Glass container
 - spent etch solution (reuse)
- PCB Drill Set
- Pin Vise or Low run out Drill Press
- Caliper – Digital very useful

Etching Solution

HCL/Muriatic Acid

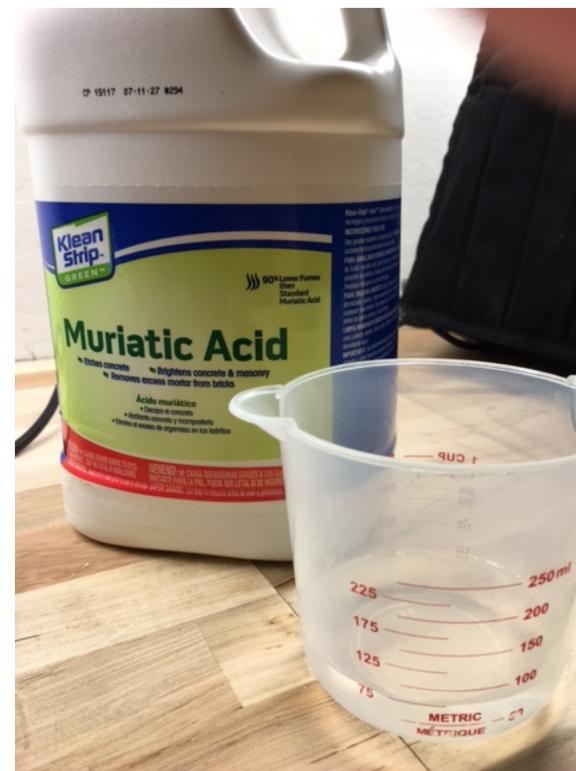
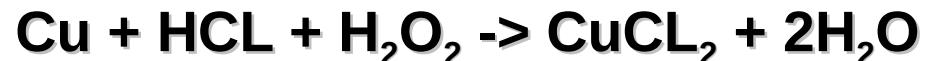
- Pool Supply or Home Depot
- Gallon more than enough for many years

Hydrogen Peroxide

- Must be clear solution
- 30% to 40% strength optimal. Works with 2% to 3% from CVS, but takes a lot longer to etch

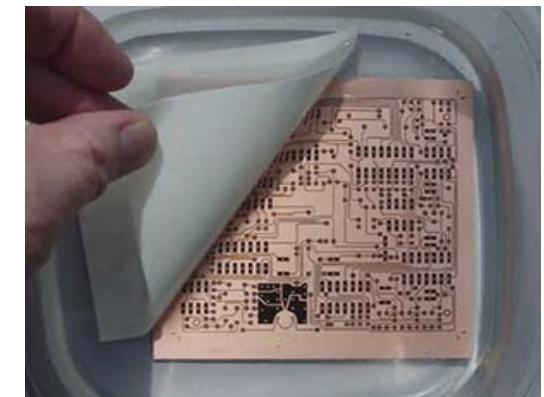
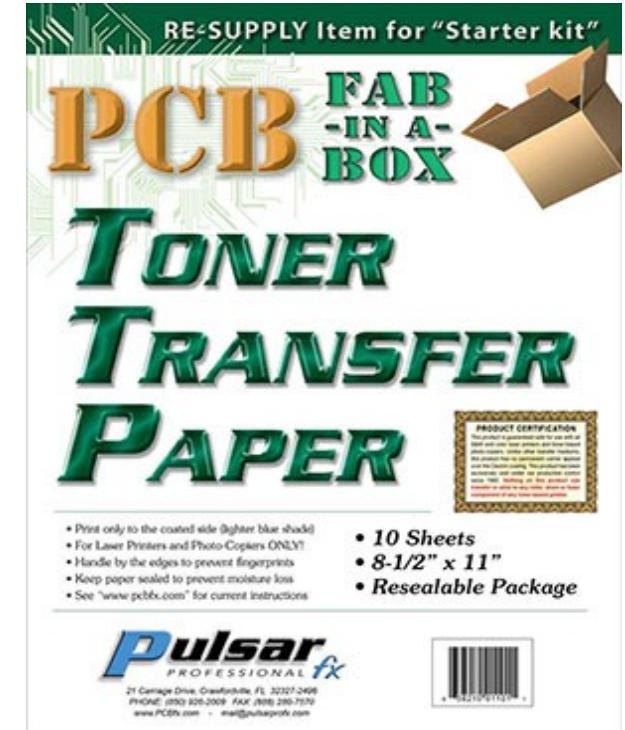
1:2 (or 1:3) HCL:H₂O₂ Ratio

75mL HCL and 150mL of H₂O₂



Magic Paper

- PCB Fab in a Box
- Dextrose coated Paper
- Optimal for Toner Transfer Process
- Full 100% transfer to copper
- Does not leave residue to hamper etching
- Great **site** with many useful links and references



Apache 18P Laminator



- Forget the clothes Iron
- Works w/ 1/16th, 3/32nd" boards
- No hardware modifications necessary
 - Some slow down roller speed and increase roller pressure
 - I just use more passes
- 375F high temp plenty hot
- Pre-Heat 10m prior to using
- Set heat to 0F and cool down 20m so rollers don't get a burned side before turning off

Drilling Thru-Holes/Via's

- Need low run out drill chuck
- Collection of PCB drill bits
 - Buy a set, #30 thru #80
 - .019", .026", .031" etc.
 - Break very easily
- Can be done w/ dremel if careful
- Use soft wood to drill into vs. hardwood.
 - Pine ok, Poplar breaks bits.



Cutting PCB's

- Can cut 3/32" w/ scissors or box cutter
 - Score then break
- Light work w/ a small bandsaw for thicker boards
 - 1/8th dual sided
- Should work with paper cutter



Work Safely

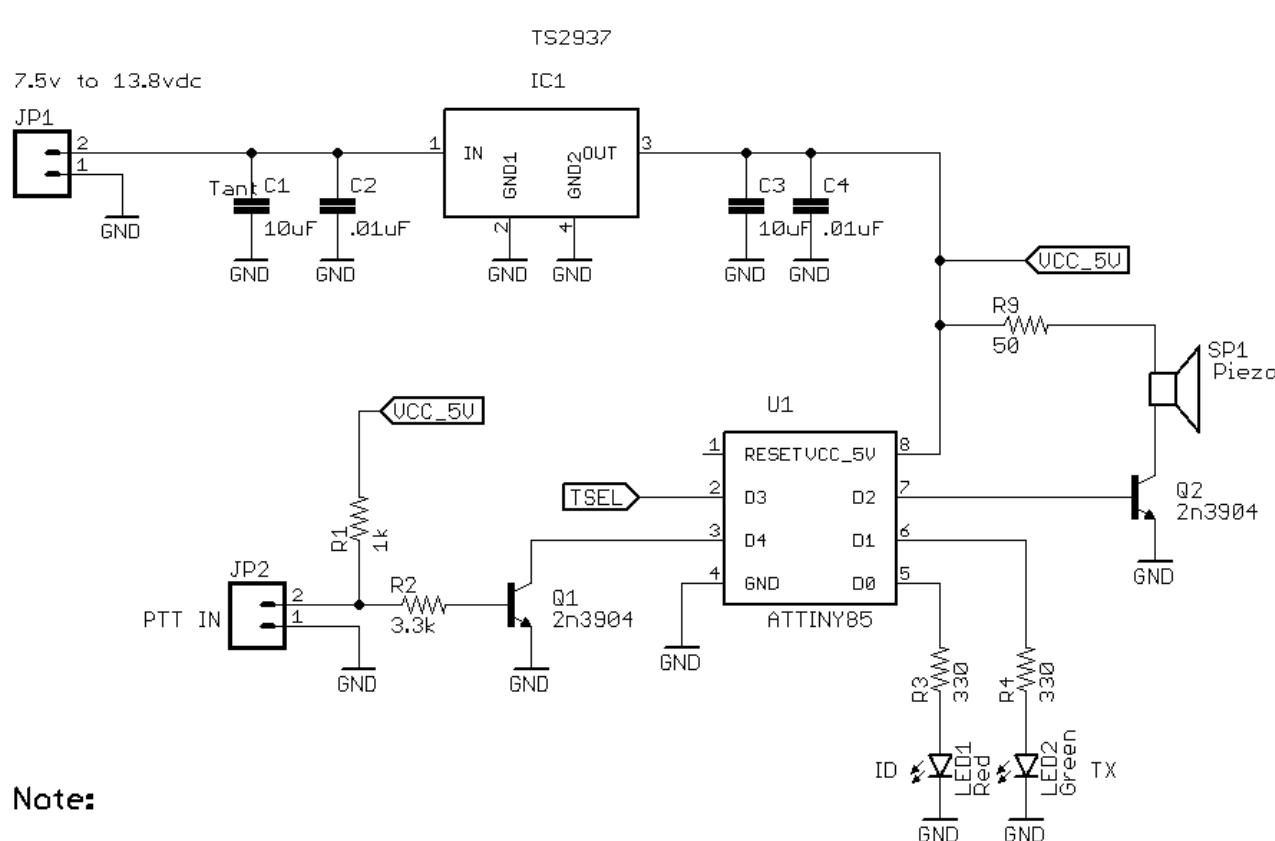
- Clean Open Space
 - Not kitchen table
- Wear Gloves / Goggles
 - Mask when sanding boards
- Have spare water readily available
- Pour H₂O₂ in first then add the Acid slowly
 - Use measuring cup(s)
 - 1:2 or 1:3 ratio acid:peroxide, so doesn't get too hot
- Paper Towels on hand
 - Chemicals
 - Paper (keep dry)
 - Peroxide (keep dark)
- If you use Acetone or ISO-Propol Alcohol
 - let towels dry completely before throwing away!
- Do not re-use Pyrex in kitchen
 - Better safe than sorry.

Workspace

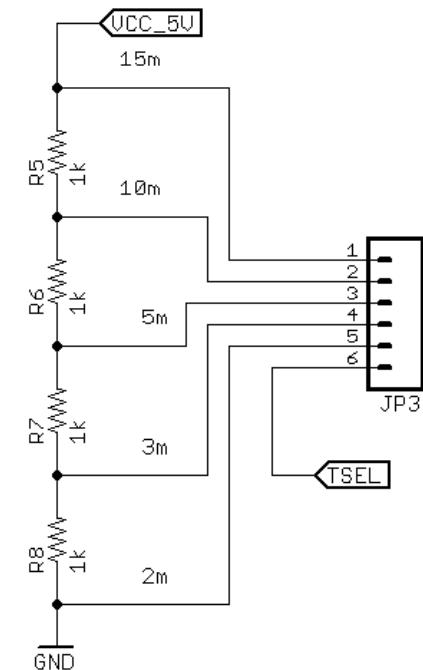


The Basic Steps

- Schematic Capture
- Board Layout/Routing
- Printing
- Board Prep
- Lamination
- Etching
- Board Clean
- Optional: Conformal Coating
- Cutting/Drilling
- Clean UP



Timer Selection
Voltage Divider



Note:

- 1) Consider using relay or opto-isolator for PTT input
- 2) Wire JP3 to 5-pos rotary switch with TSEL to center wiper
- 3) Use .5% or 1% 0805 resistors for timer selection voltage divider
- 4) Piezo should be ~40 ohms. R9 limits current/volume.

Released under the Creative Commons
Attribution Share-Alike 4.0 License
<https://creativecommons.org/licenses/by-sa/4.0/>

TITLE: YARC_ID_Timer

Design by:

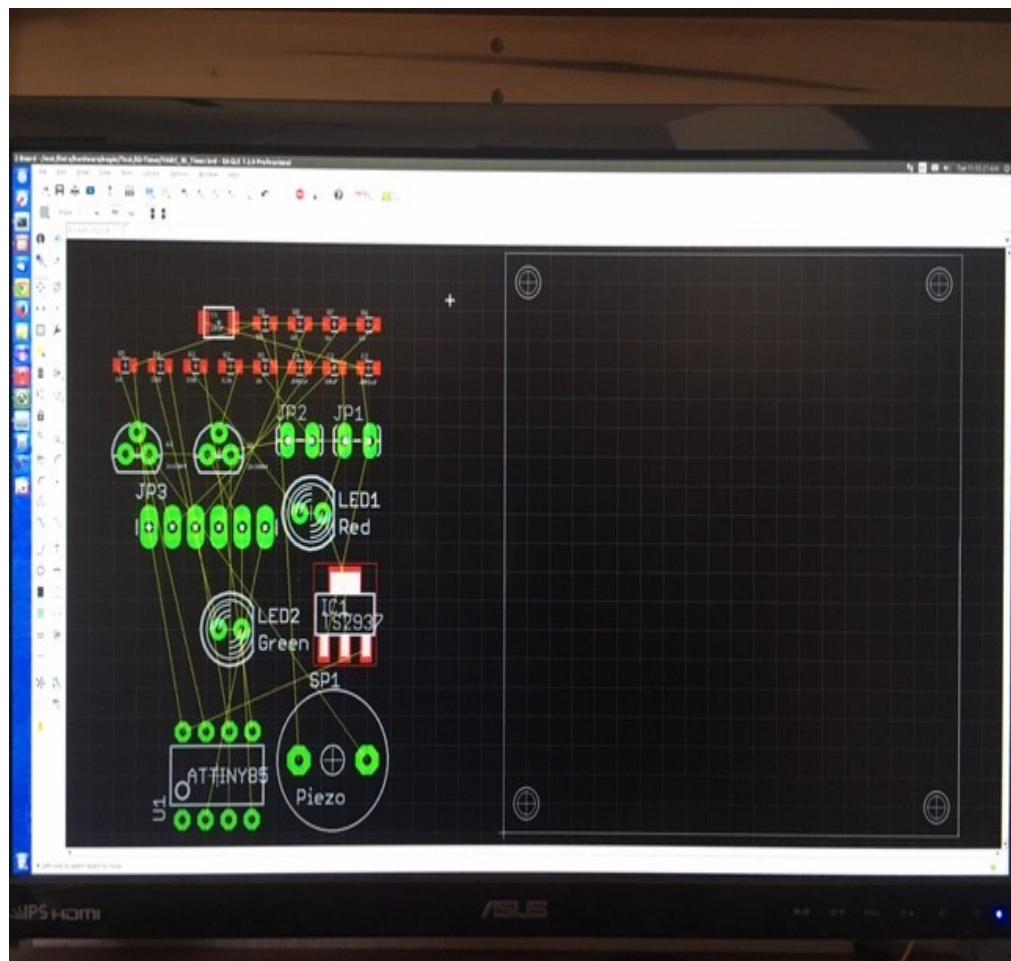
Douglas Theriault - N01D

REV:
v1.0

Date: not saved!

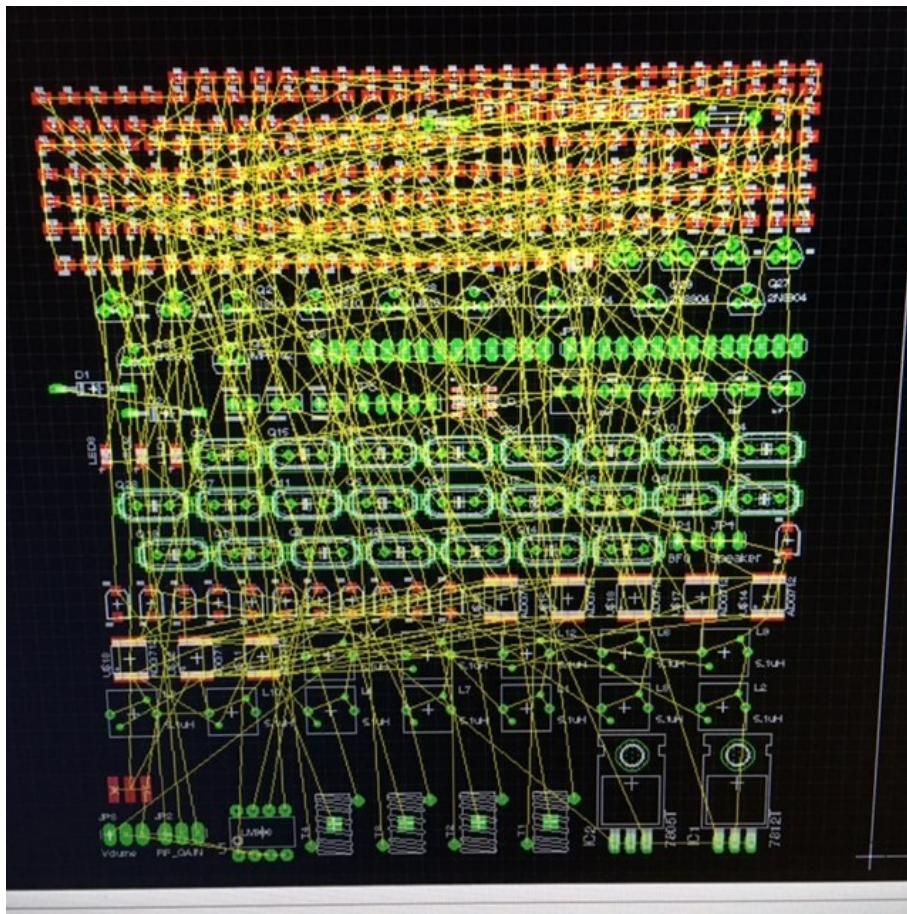
Sheet: 1/1

Routing the Rats Nest



- Drag parts from the rats nest onto your board
- Rotate, Align parts
- Realize your vision for the layout
- May or may not align with schematic layout
- Perhaps draw first on paper
- DFM (design for manufacturing) considerations apply

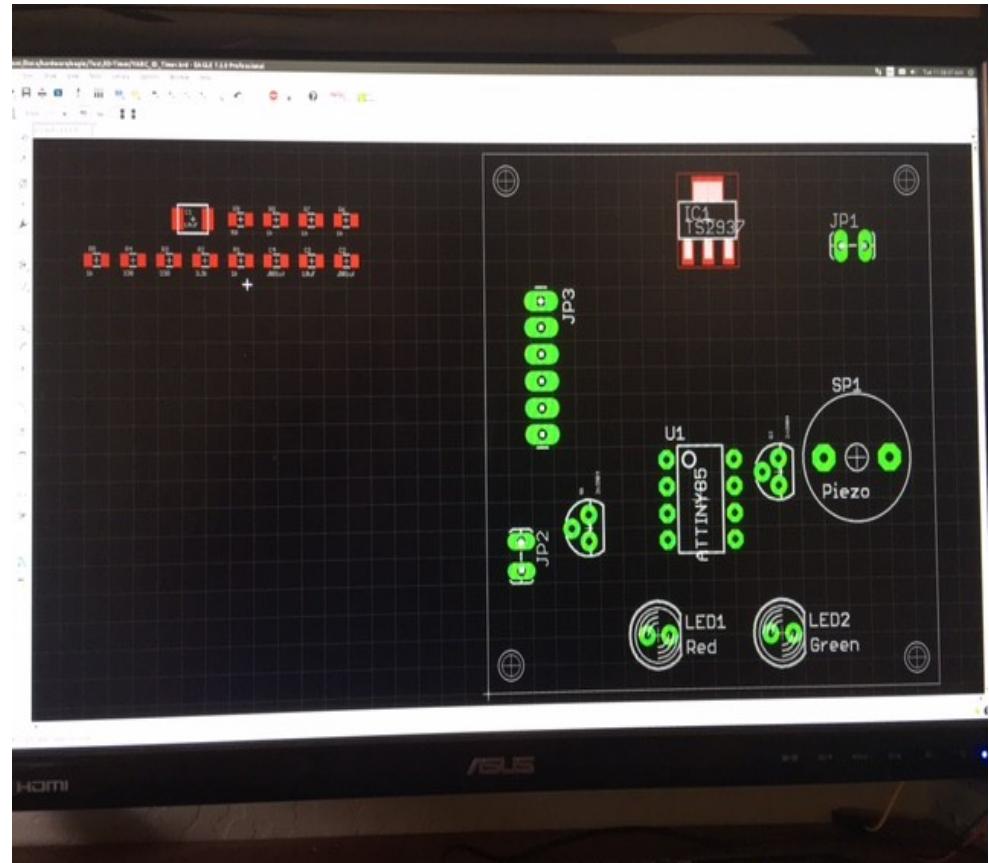
Here's one.. Teensy Rx Design...



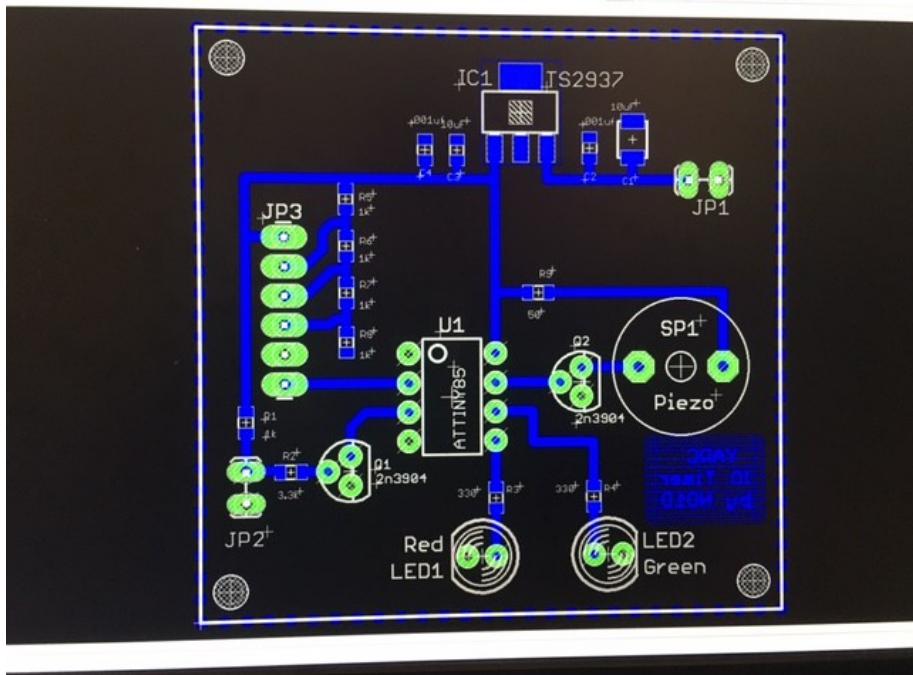
- Took several days to get through manual routing
 - Only 350 parts
 - 2 sided
 - Nice rats nest...

Trial and Error Process

- Turn off un-routed traces while you adjust layout and positioning of parts
- Find layout bit of an art form, draw on paper first to realize your vision



Routing – Adding Traces

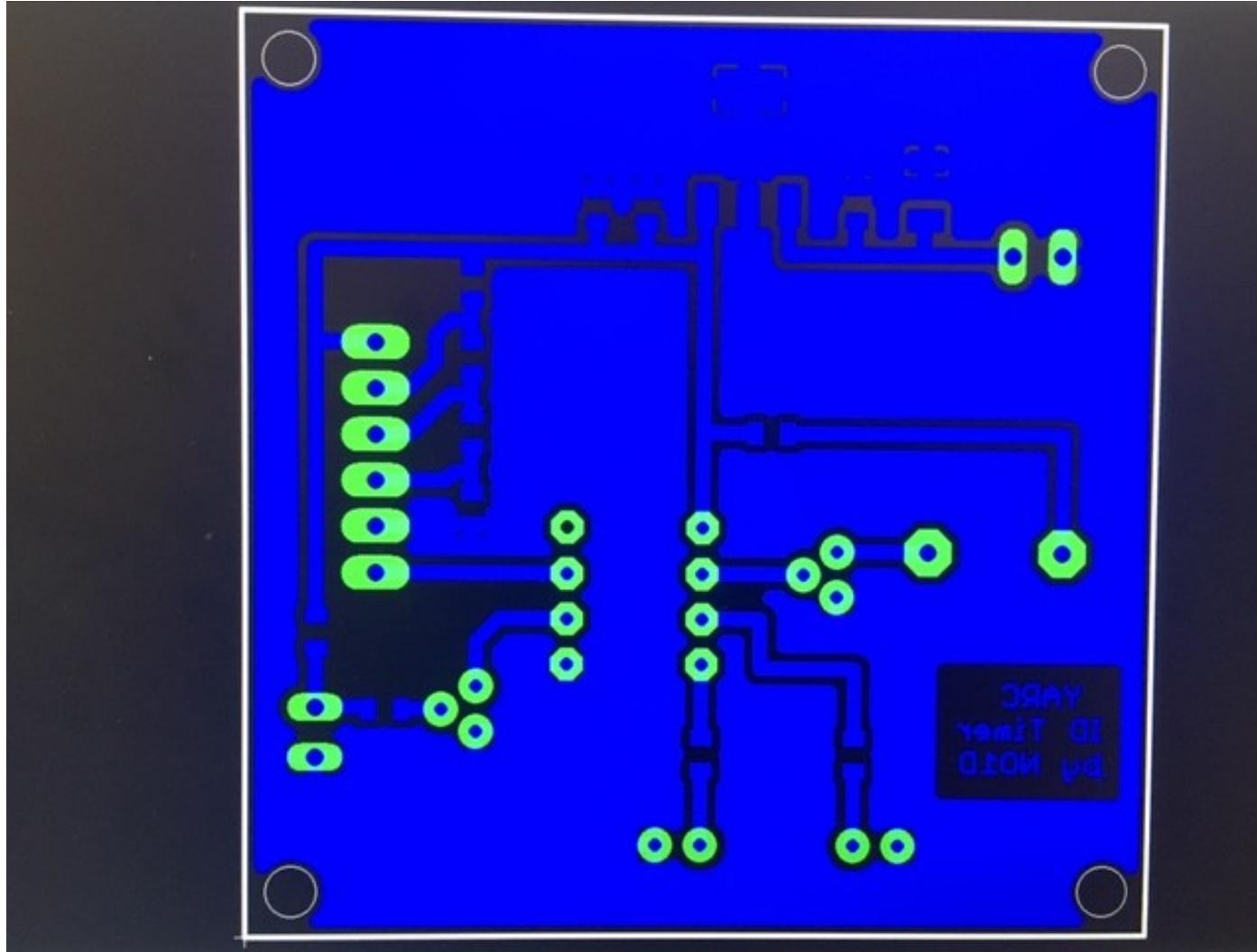


- Prefer manual routes vs. auto-routing
- Try to minimize via's (or wires)
- Adjust widths for power if needed
- Hint: Don't change grid size; .01" min. but can cause theta errors with traces if you change grid spacing often

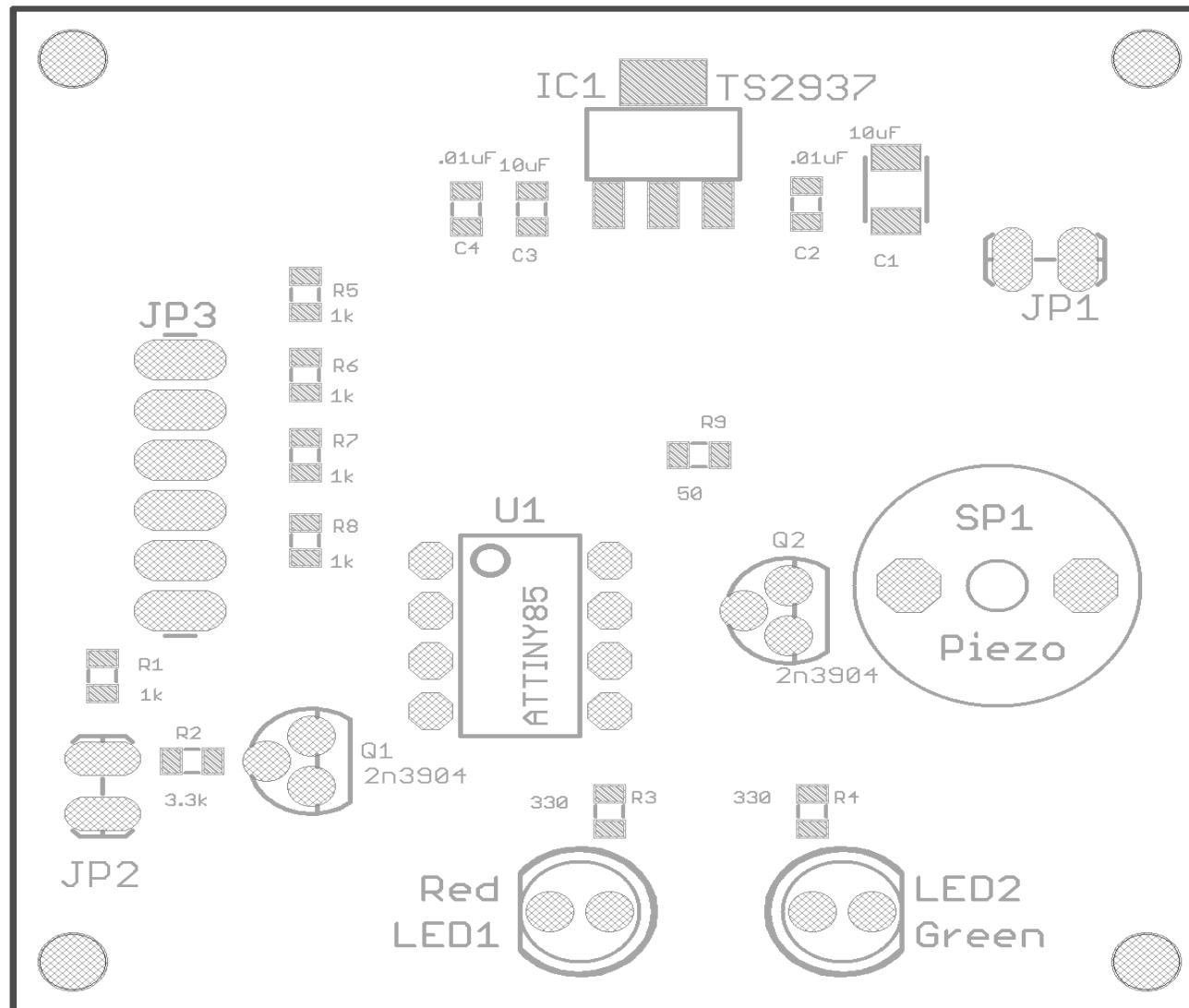
Routing, Board Layout

- Set dimension of board size
- Add mounting holes
- Connectors and External facing parts first
- Organize parts similar to schematic
- Turn off un-routed lines initially to orientate/move parts on board
 - less confusing on high density boards
- For 2 layer boards, take advantage of both sides for SMD components, helps reduce via counts
- Manually routing Vs using “auto-routers”
 - Better layouts/control over paths. Less reworking...
- Add ground plane(s) to top and or bottom layers
- RF Transmission lines possible w/ 2 layers however not easy interfacing with SMD pads. Traces typically too wide.
 - Works best w/ 4 layers but not in your garage.
 - Great tutorial by Mike Ossman AD0NR on YouTube: Simple RF Circuit Design
 - https://www.youtube.com/watch?v=TnRn3Kn_aXg
 - Lots of online calculators to figure out trace widths given type of PCB and desired impedance

When you're Done... Bottom Layer w/ Pads and Via's



Silk Screen Layer

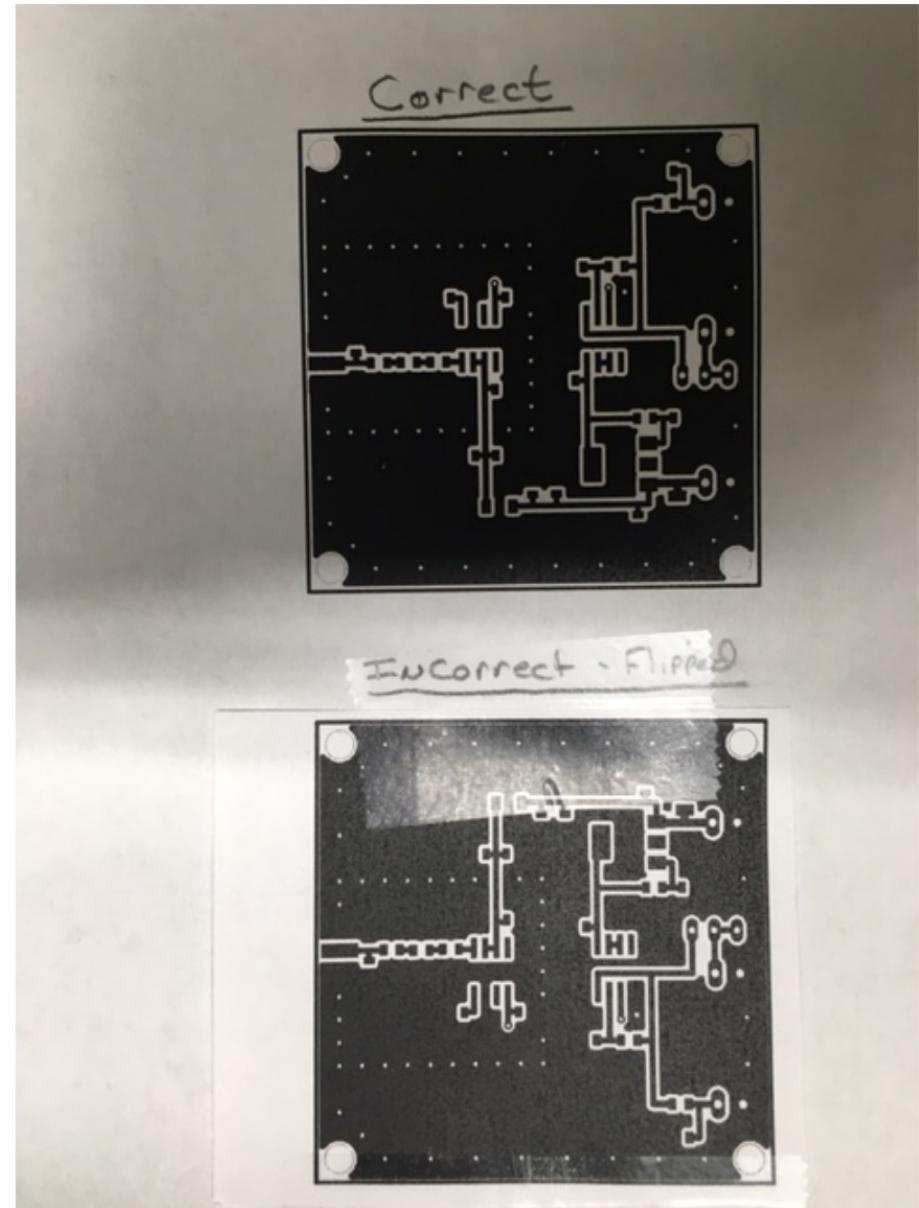


Printer Settings

- Black, Solid print
- 600 dpi, 1200 or greater is better
- Select thickest Paper (Bond)
- Graphics Quality
- No Toner Saving
- Printer performance varies
- Best results with a fresh toner cartridge

Double Check Print

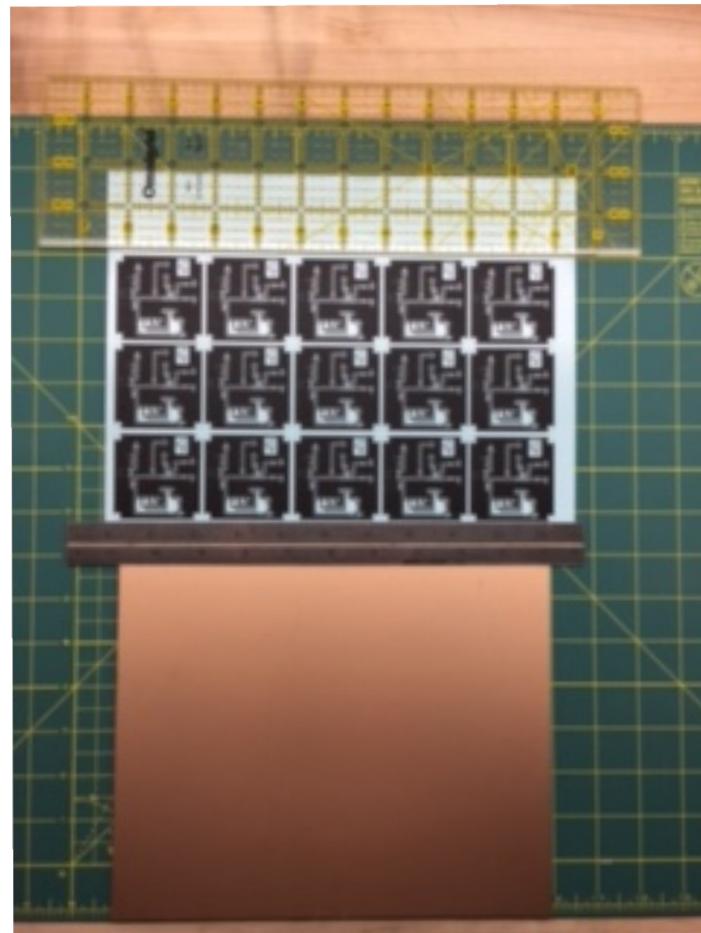
- Print on normal paper first (or transparency)
- **Top Layer Mirrored**
- **Not the Bottom Layer**
- Check Part footprints before committing to final paper
 - Digital Caliper very useful



Hints, Tricks and Gotchas

- This Ol' PCB
 - Measure Twice, Cut Once
- Check packages
 - Proper part sizes !
- Printing
 - **Top Layer Mirrored**
 - **Bottom layer NOT !**
- Watch out for Scaling factors
 - Print to .pdf sometimes scales
- Print layers to .ps files then combine using image editor to create panels,
 - uses less dextrose paper

Print onto Dextrose Paper



Board Preparation

- Wash copper boards using soap, green scrub pads
- Removing the oxide layer
 - Board turns from orange to pink in color
- Remove all oils from surface.
- Hold by board edges to keep finger prints off board
- Final cleaning/drying, use Iso-Propol Alcohol.



Toner Transfer

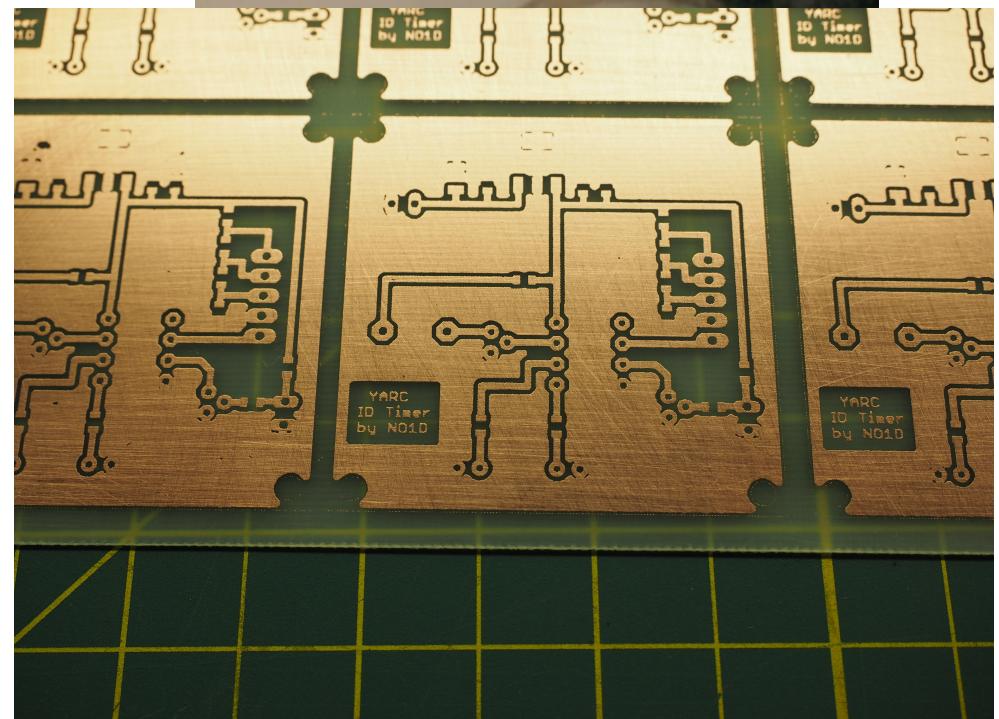
- Transfer Toner onto Copper board via Lamination
- Trim printed paper to fit board
- Use blue-tape if needed to stabilize
- Pre-Heat Laminator; 10m ok
- Use water to separate paper from PCB
- Application of Conformal Coating
 - Optional Step
 - Used to prevent pitting of copper during etching

Watch the Process

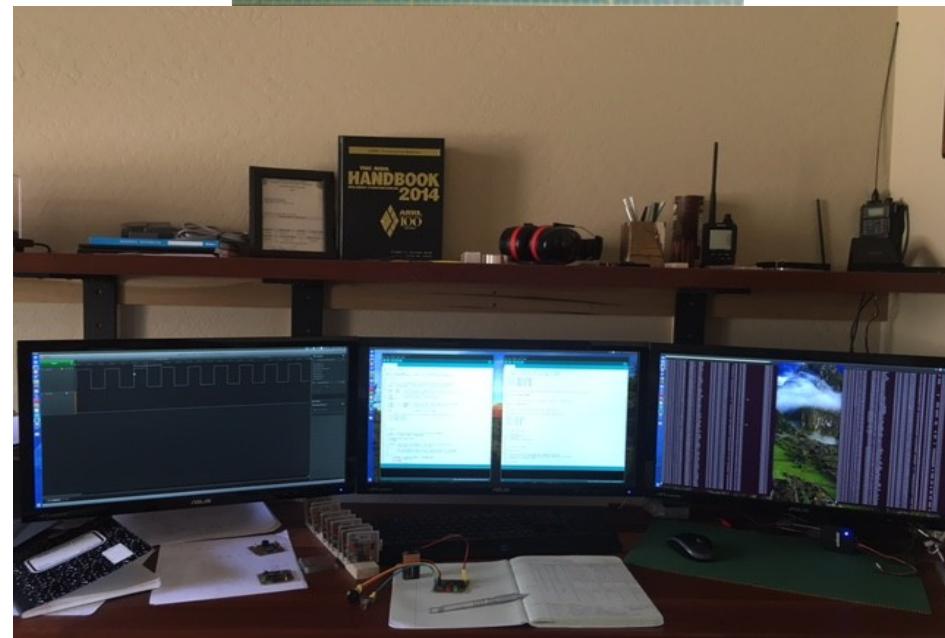
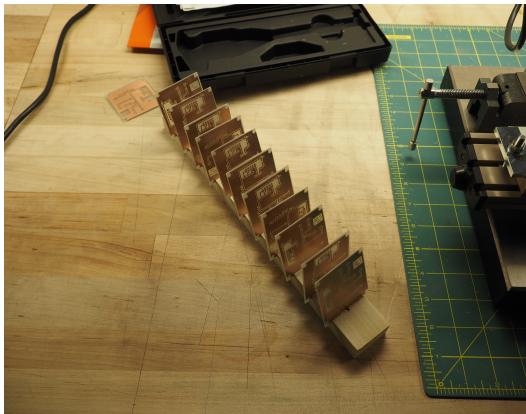
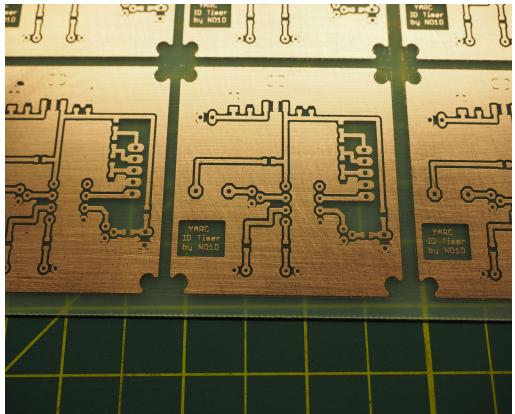


After Etch Cleanup, Remove Toner

- Can be done with Acetone
 - Or Thinner or Nail Polish Remover
- Recommend using Green Scub Pads, water and elbow grease... Less Flammable..
- Wipe down with Alcohol
- Then check etch results. Look for broken traces or incomplete etching.
 - Close inspection may reveal jagged edges on traces, possibly due to printer (dpi)

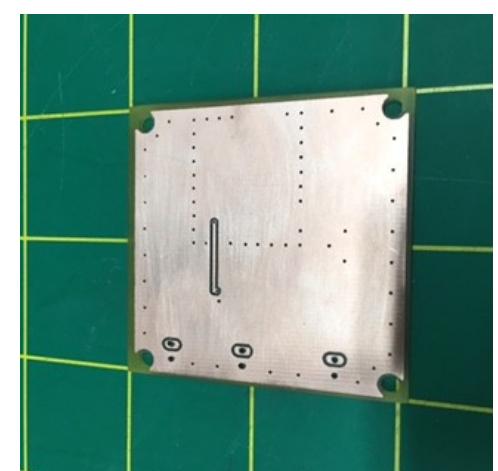
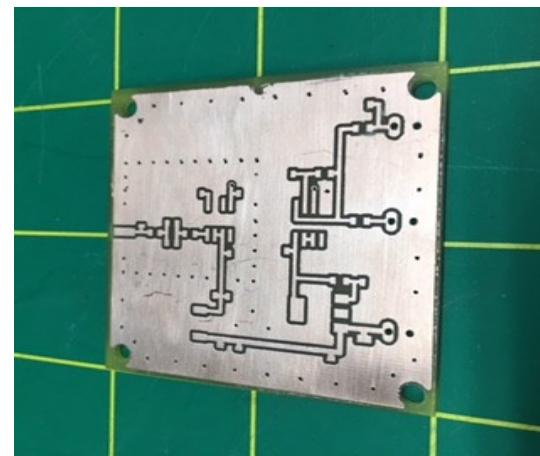
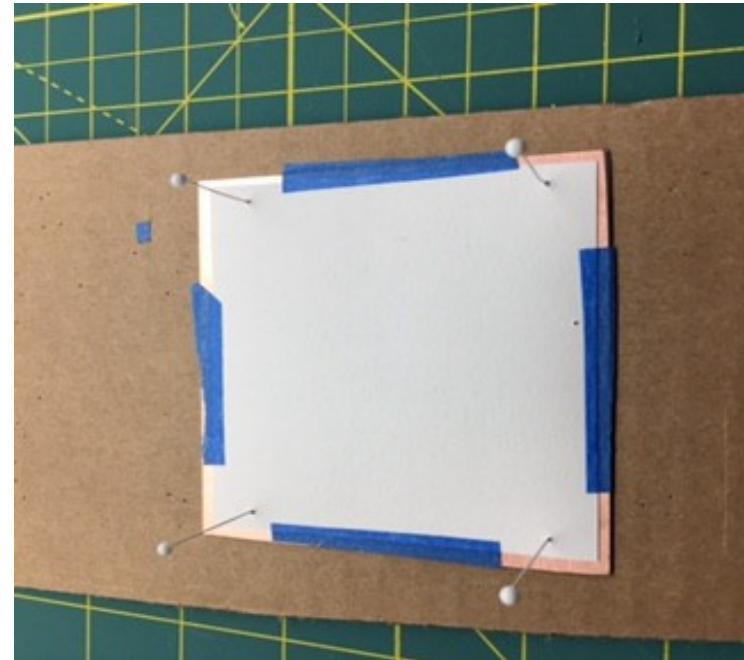


Cut, Drill, Assemble



2 sided PCB's are Possible

- Prep 1st side as with single layer board
- Prior to Etching, protect 2nd side using Contract Shelf Paper (plastic)
- Etch 1st side & clean, remove contact paper
- Drill (4) registration holes, dia of sewing pin; use (4) corners, via holes
- Insert pins into transfer paper, then into PCB; over piece of cardboard
- Blue Tape transfer paper while pins are inserted
- Finally protect 1st etched side with contact paper
- Perform 2nd etch & clean...



Links/References

Paper	http://www.pcbfx.com/main_site/pages/products/transfer_paper.html
Toner Foils	http://www.pcbfx.com/main_site/pages/products/toner_foils.html
Laminator	http://www.amazon.com/Apache-AL18P-Professional-Laminator-Documents/dp/B0012UKI72
H2O2	http://www.amazon.com/Salon-Care-Volume-Clear-Developer/dp/B004OKDW5W
Eagle PCB	http://www.cadsoftusa.com/
KiCad	http://kicad-pcb.org/
Arduino	http://www.arduino.cc
ATtiny85 Datasheet	http://www.atmel.com/images/atmel-2586-avr-8-bit-microcontroller-attiny25-attiny45-attiny85_datasheet.pdf
Tiny AVR Programmer	http://www.sparkfun.com/
YARC GitHub Repository	Software, Board Files, Design Document, Presentation can be downloaded from GitHub. https://github.com/dtheriault/YARC
Questions?	Email me at: no1d.doug@gmail.com