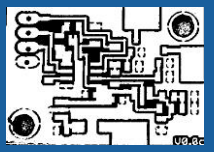
A Recipe for Rapid Printed Circuit Board Prototyping

Wayne Getchell
VE3CZO

What's Covered in the Talk



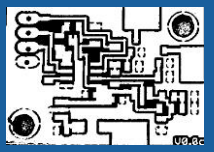
PCB design tools

- Component libraries
- Circuit schematic capture
- PCB Layout
- Tooling artwork generation - Gerber files

Homebrew PCB fabrication process to create single or double sided boards in a few hours

- Line and spacing limits
- Artwork generation
- Direct contact printing
- Etching
- Plating
- Final cutting, trimming, and vias

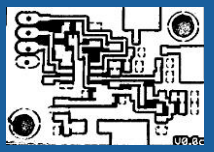
PCB Front End Tools



Two software packages for schematic capture, board layout, library parts generation, and auto routing

- Eagle
  - Light (free)
    - ≈ Limits - Single page schematic -2 layers -100x80mm board
  - Hobbyist ($169 in 2012)
    - ≈ Limits – 99 page schematic – 6 layers – 100x160mm board
  - Info & downloads www.cadsoftusa.com
- DipTrace
  - Lite (free)
    - ≈ Limits 300 pads & 2 layers
  - Non Profit (selling a few boards for your hobby is ok)
    - ≈ Standard $125 (in 2012) – Limit 1000 pads 4 layers
    - ≈ Extended $248 (in 2012) – Limit 2000 pads 6 layers
    - ≈ Full $348 (in 2012) - unlimited
  - Info & downloads www.diptrace.com

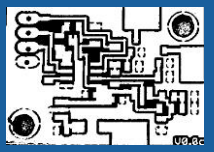
PCB Front End Tools (cont'd)



Both Eagle and DipTrace are / have

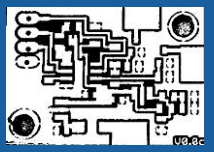
- Extremely capable but also complex and therefore not easy to use so have considerable learning curves
- In depth tutorials for most functions including library generation
- Good on-line help features
- Support forums
- Extensive libraries
- Schematic capture, board layout, trace auto routing, ground & power plane pour support, scripting for tasks like automatic parts list generation and export to Gerber format for PCB fab

Use Eagle or DipTrace?



- I use Eagle and examples in the following slides use that program.
- I haven't used DipTrace but Bert VE2ZAZ uses it and quite likes it.
- At this time I have a preference for Eagle, simply because of the time I've invested in it generating libraries and becoming familiar with the schematic and layout tools. But I must say that in investigating DipTrace I like it a lot. It also looks very powerful, doesn't have any board area limitations (although pads are limited), and has friendlier licensing terms.

Generating Library Parts



Parts are created for both schematic capture & PCB Layout

- A symbol is used to define what a part looks like on schematics
- A package is used to define what the part look like on the PCB
- A device shows how I/O pins will be connected in both

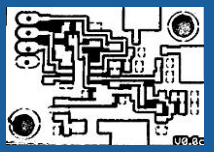
It's critical that you trust your library components so before using any part …

- Check the pin out connectivity
- Print a package at 1:1 and place the component on the printout

Why create custom components?

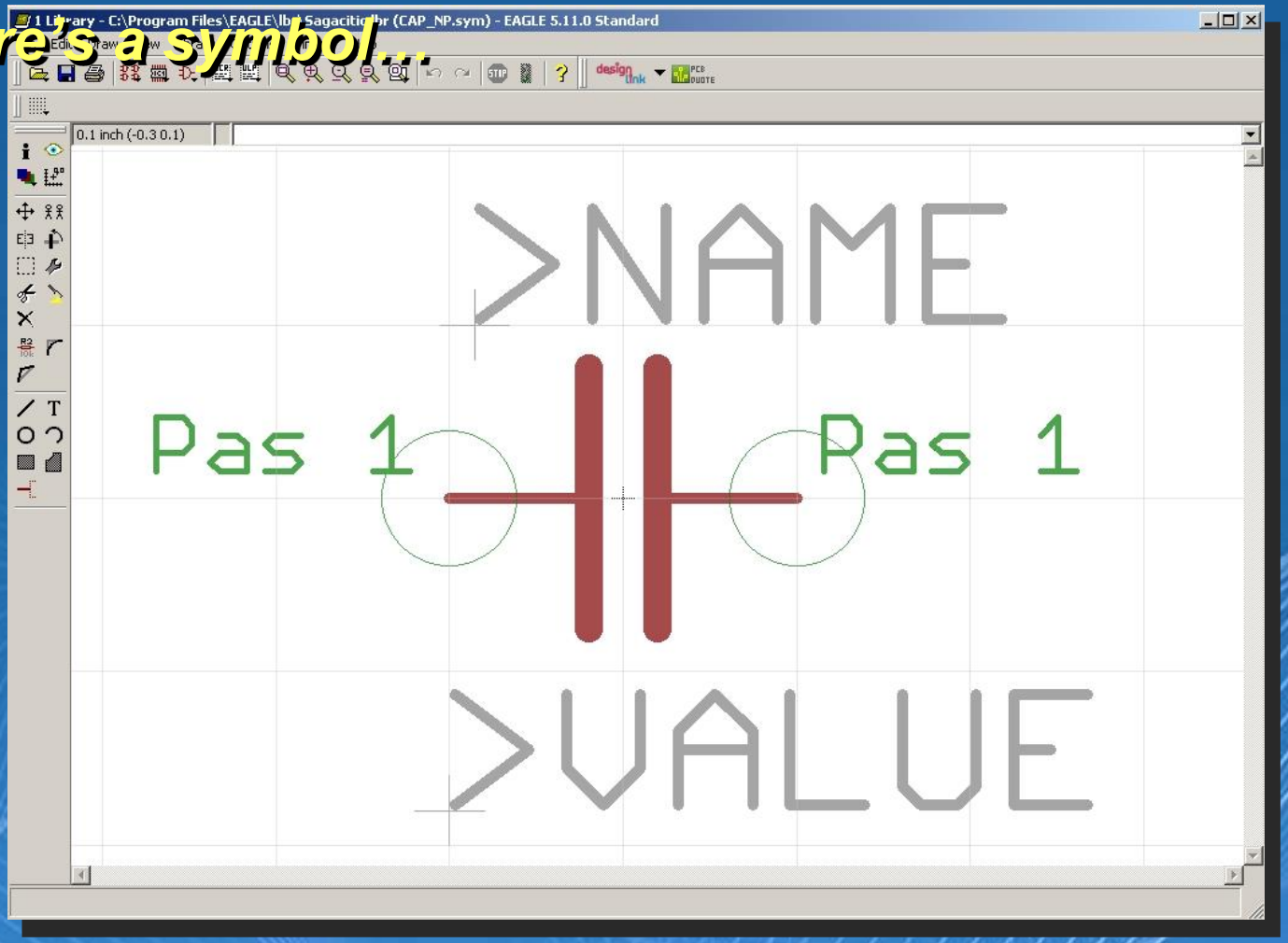
- The part can't be found in any library
- Special clearances are needed for the fab process & soldering
- Consistency of part attributes, value, part number size

Making a Library Component



Three steps: Define symbol, package, then a component

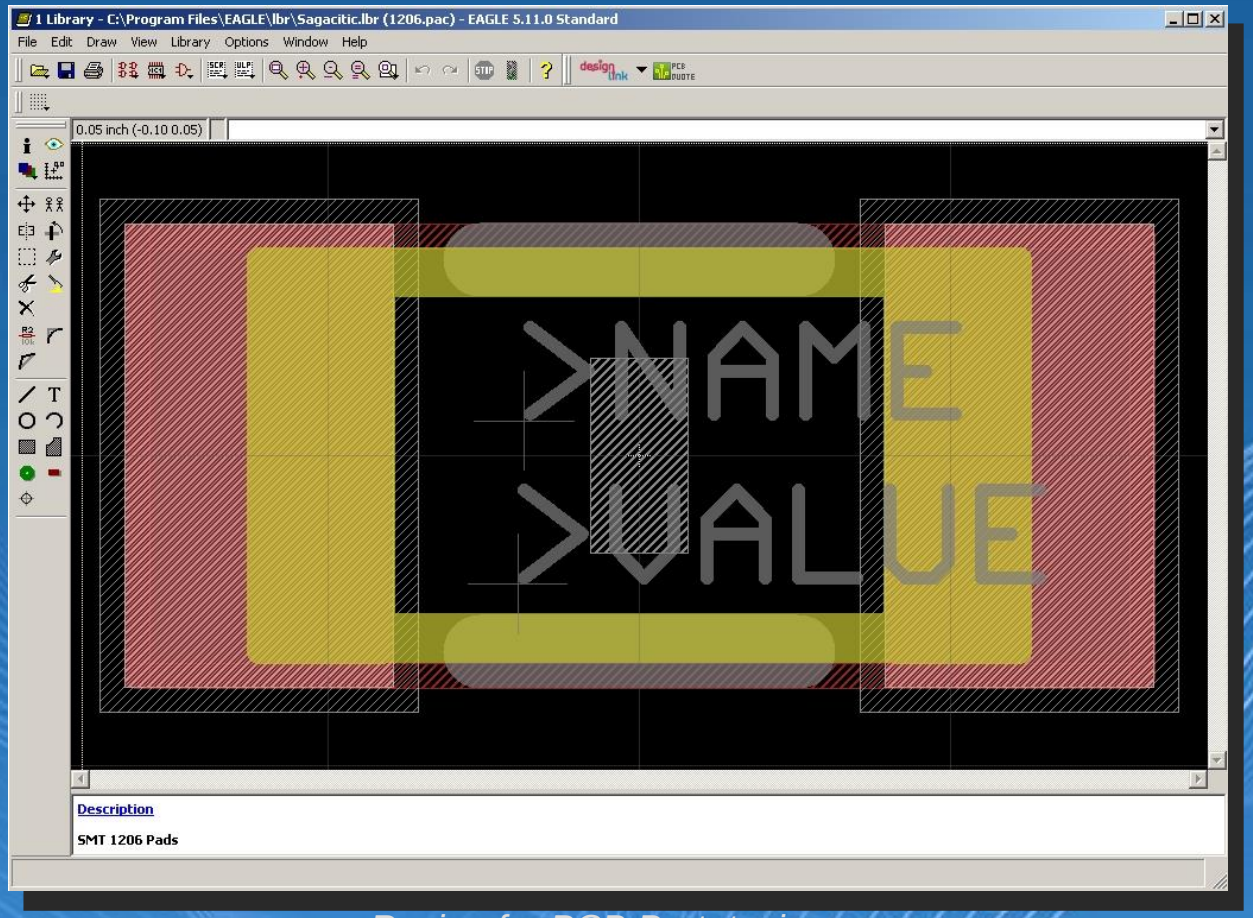
Here's a symbol…



Library part generation

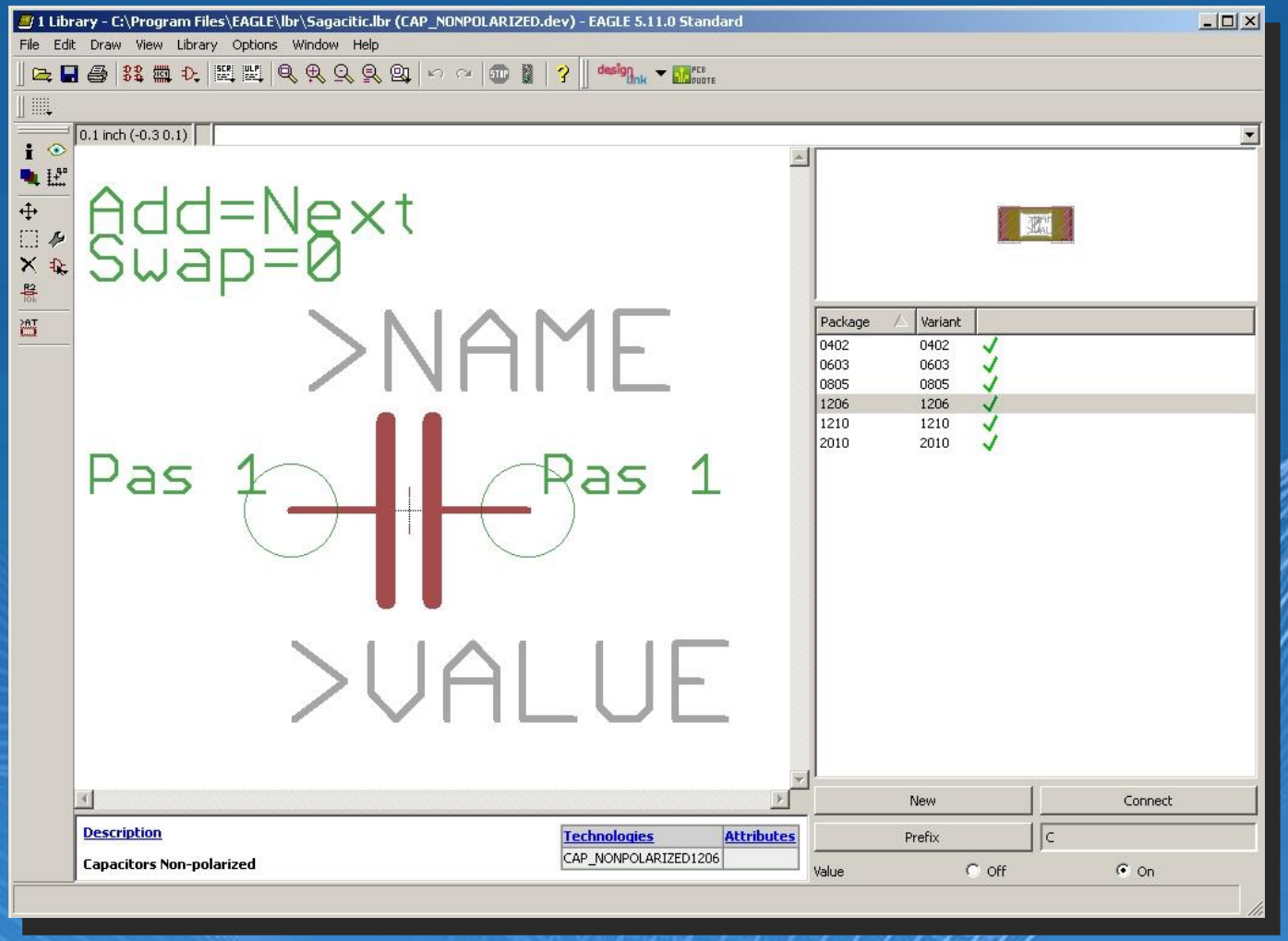
A Package…

- I tend to make pads bigger than many footprints ask so that I can hand solder to them

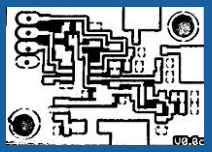


Library Part Generation

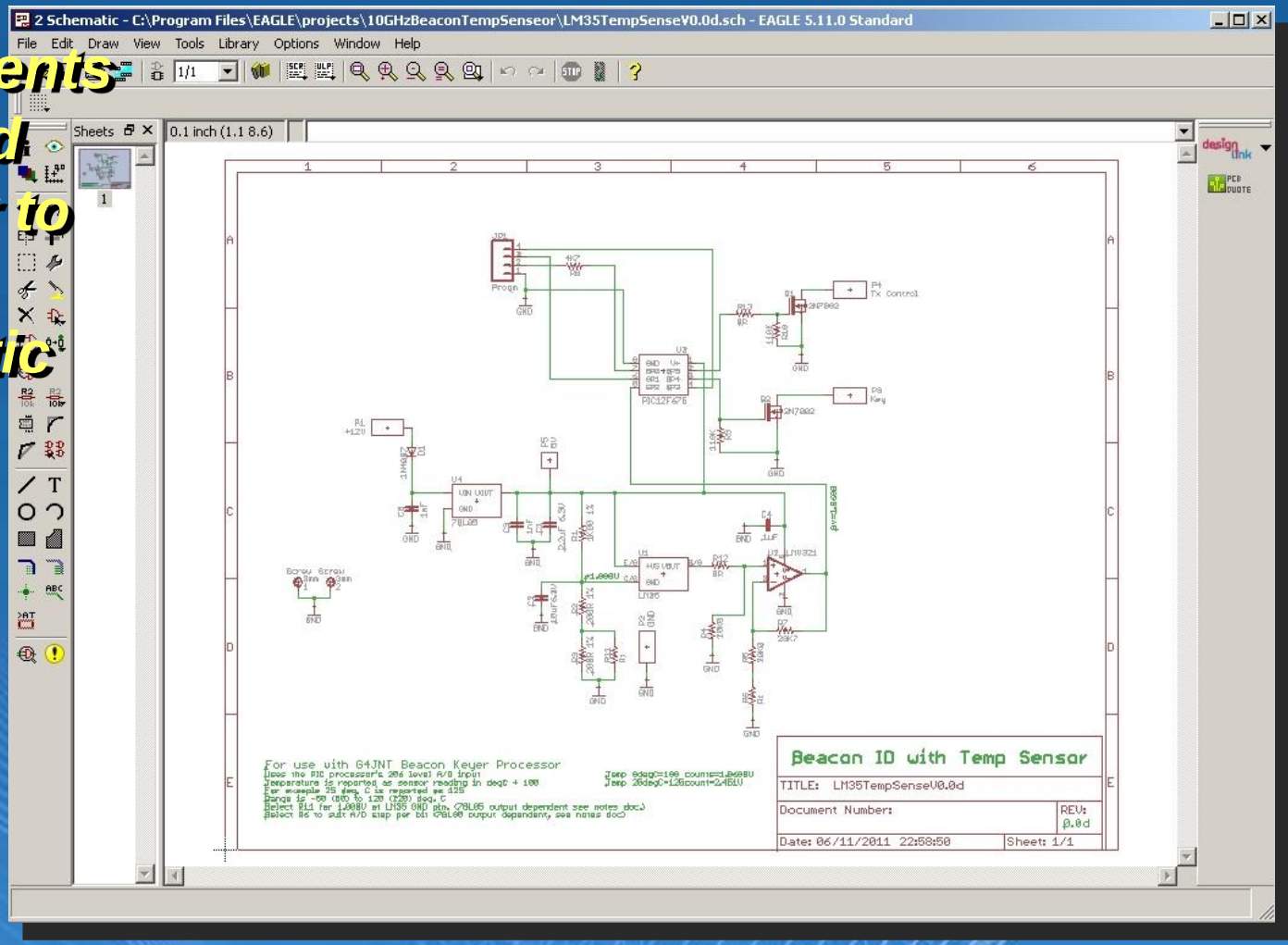
Both Symbol and package are needed for a Component



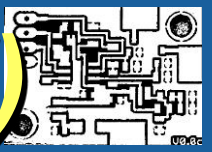
Schematic Capture



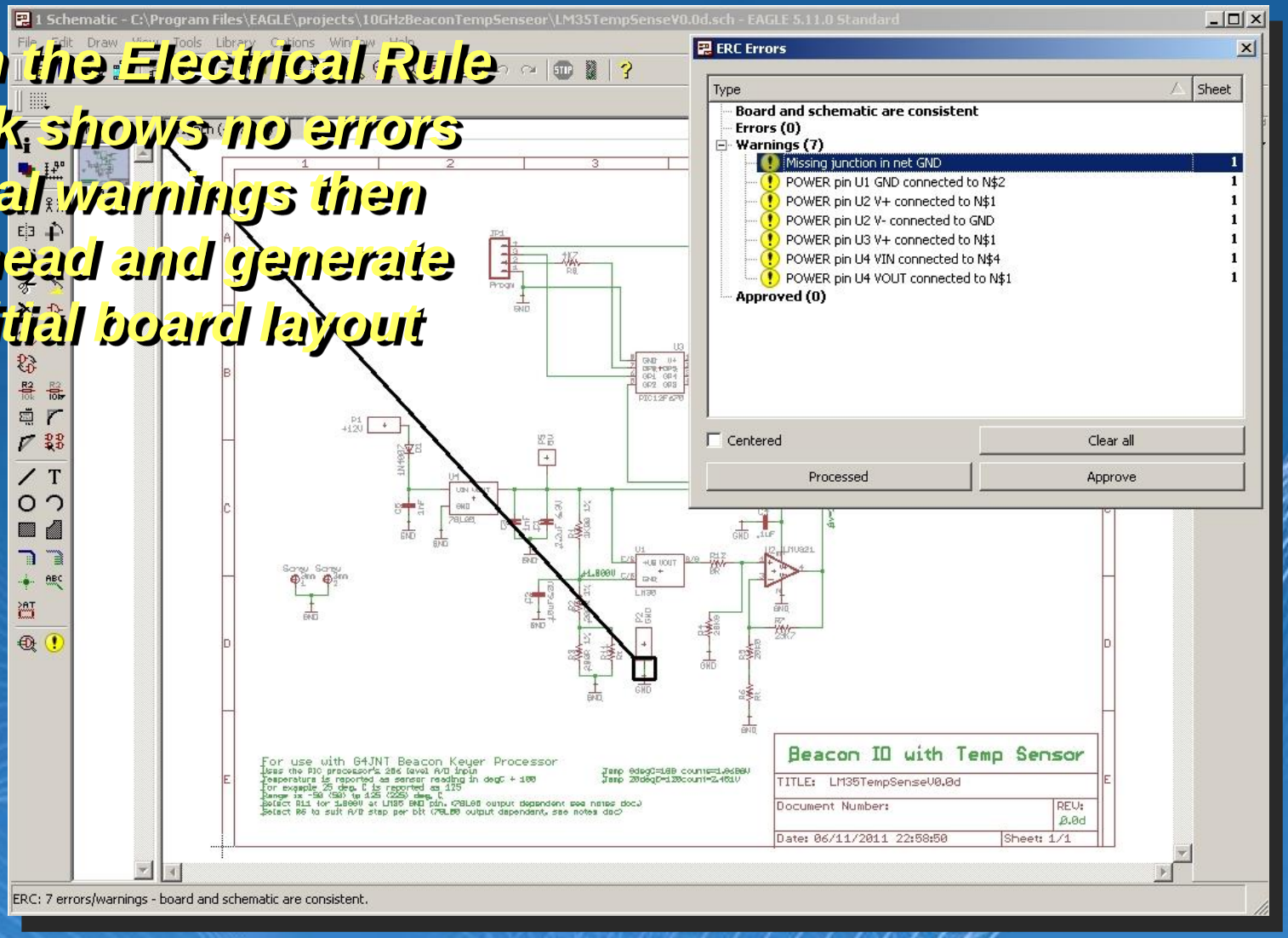
Library components are wired together to create a schematic



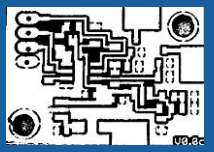
Schematic Capture (Cont'd)



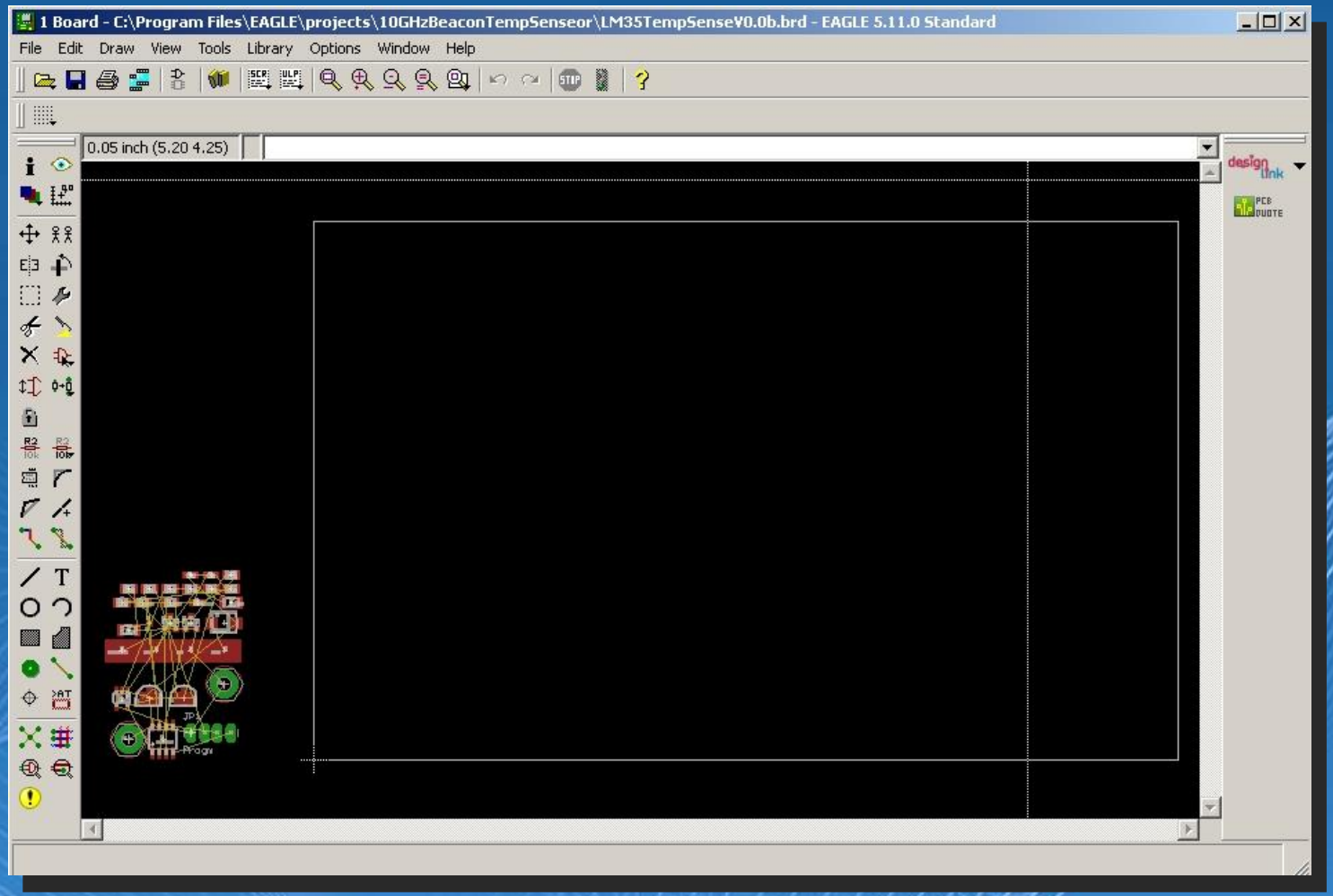
When the Electrical Rule Check shows no errors or fatal warnings then go ahead and generate an initial board layout



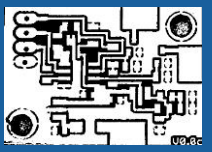
PCB Layout



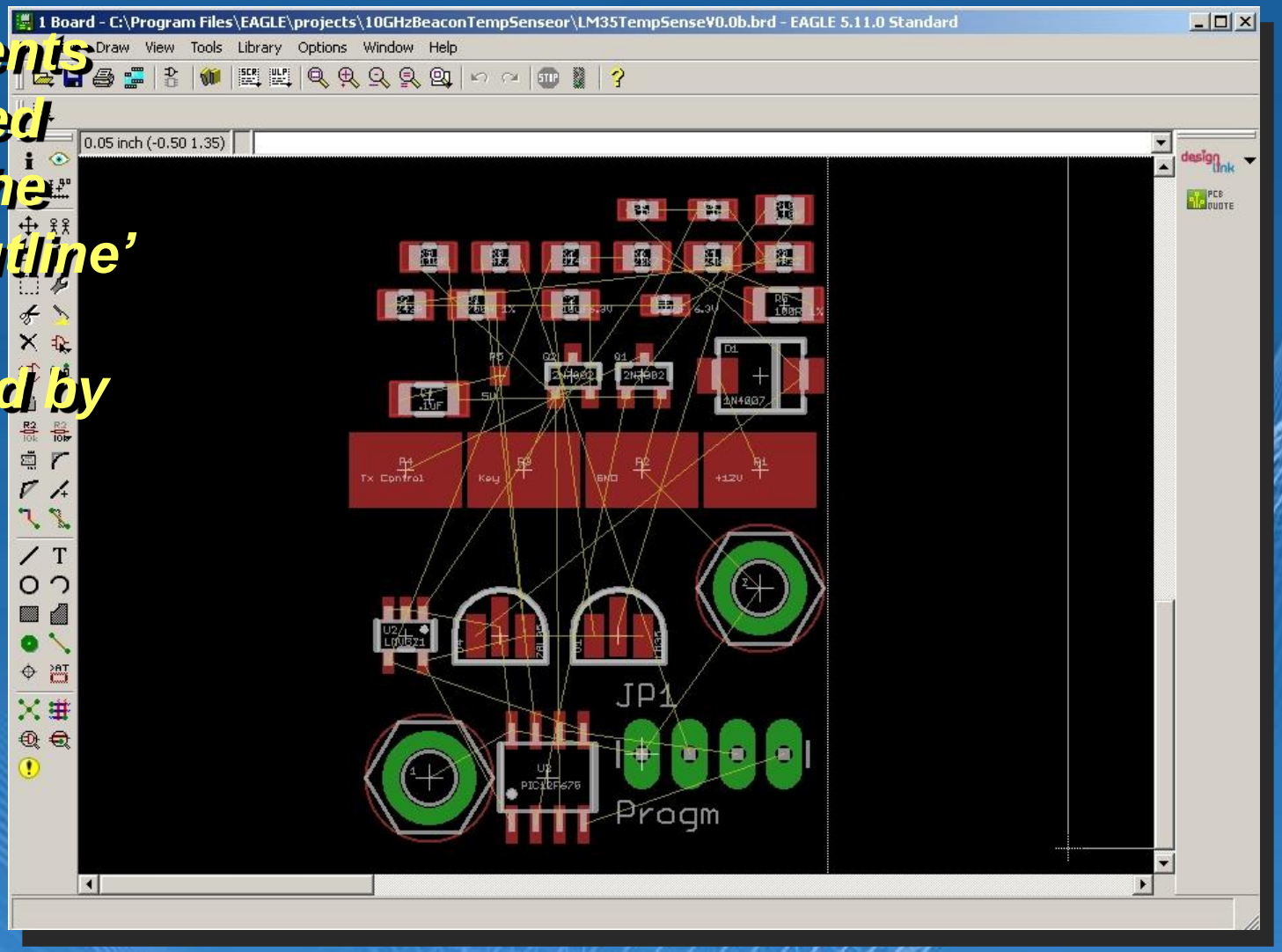
Initial Parts Layout created from schematic



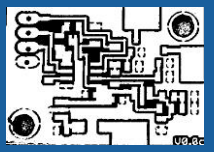
PCB Layout



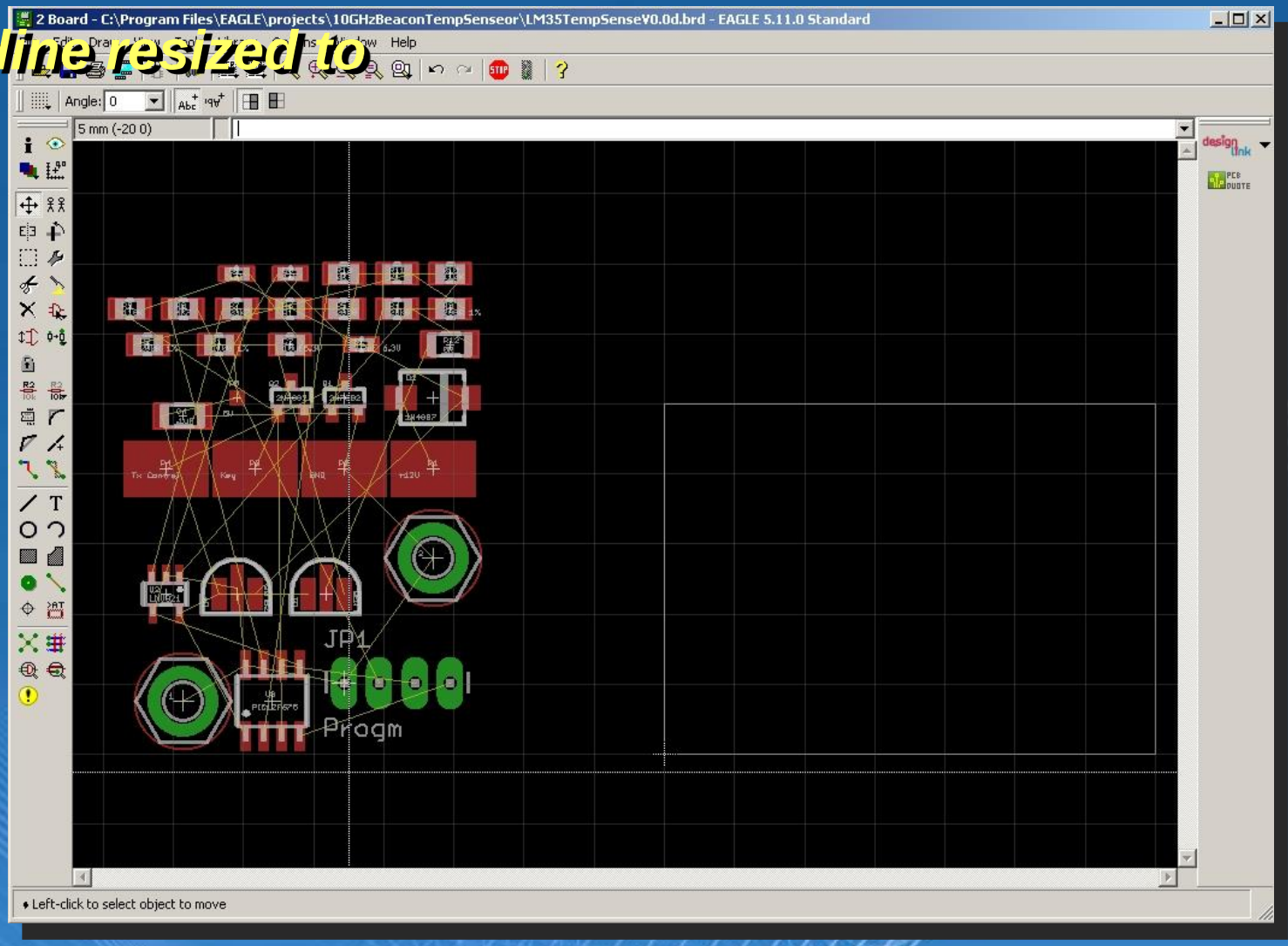
Components are located outside the board 'outline' and are connected by airwires



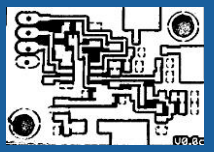
Layout



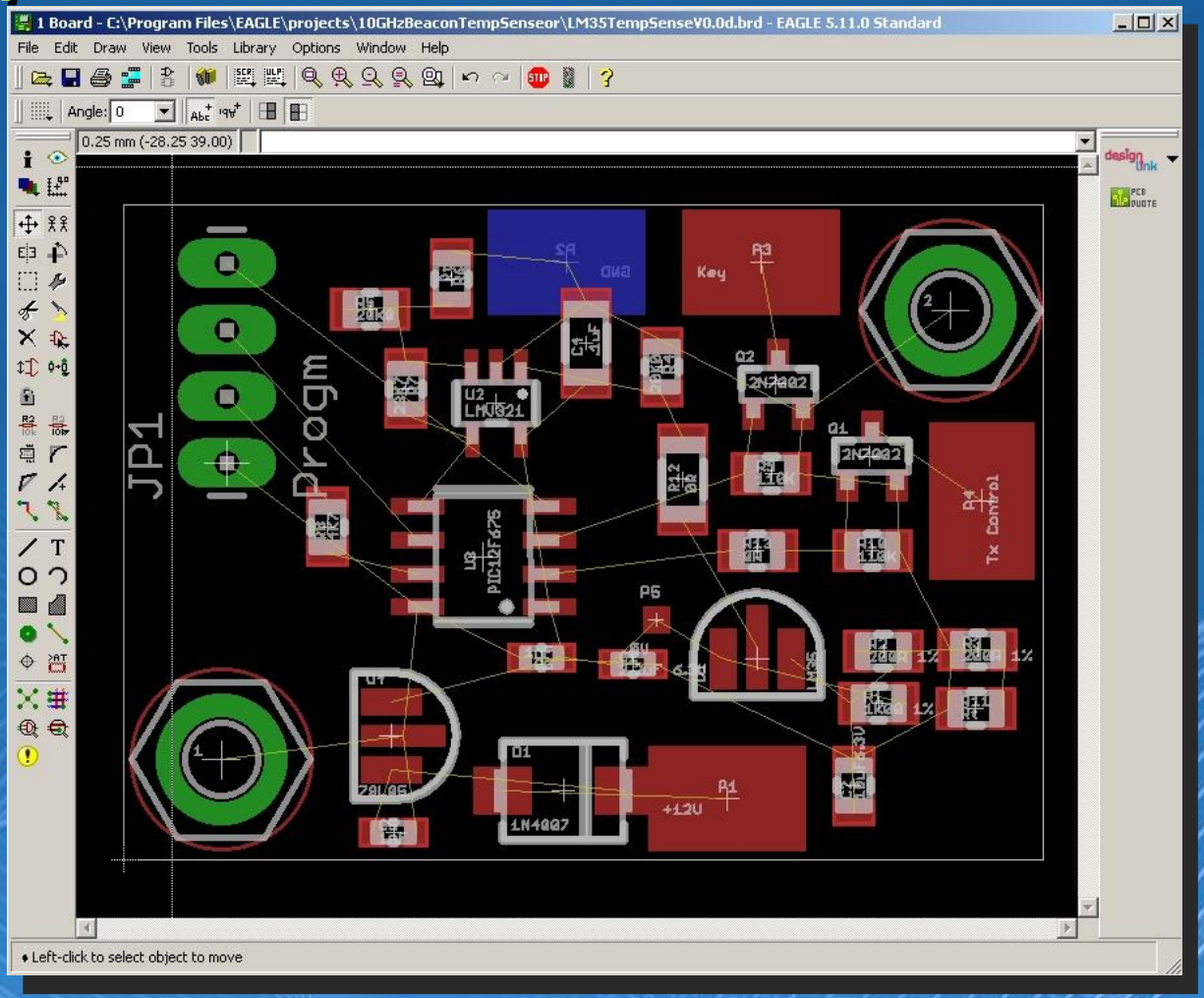
Board Outline resized to 25x35mm



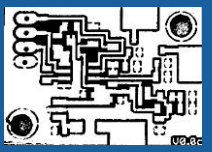
PCB Layout



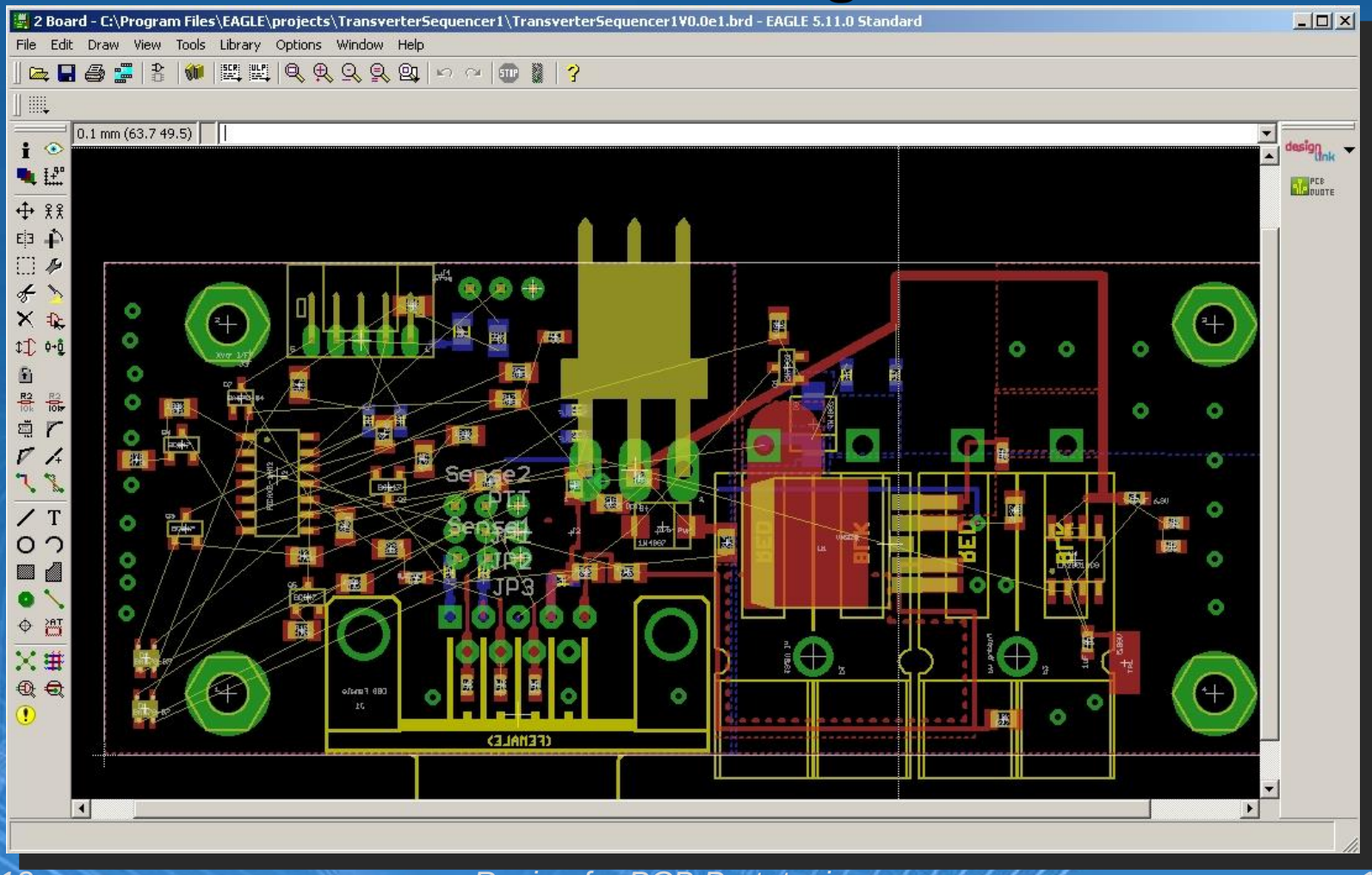
Component Placement



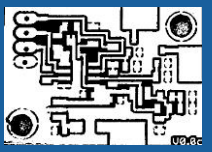
More AirWires



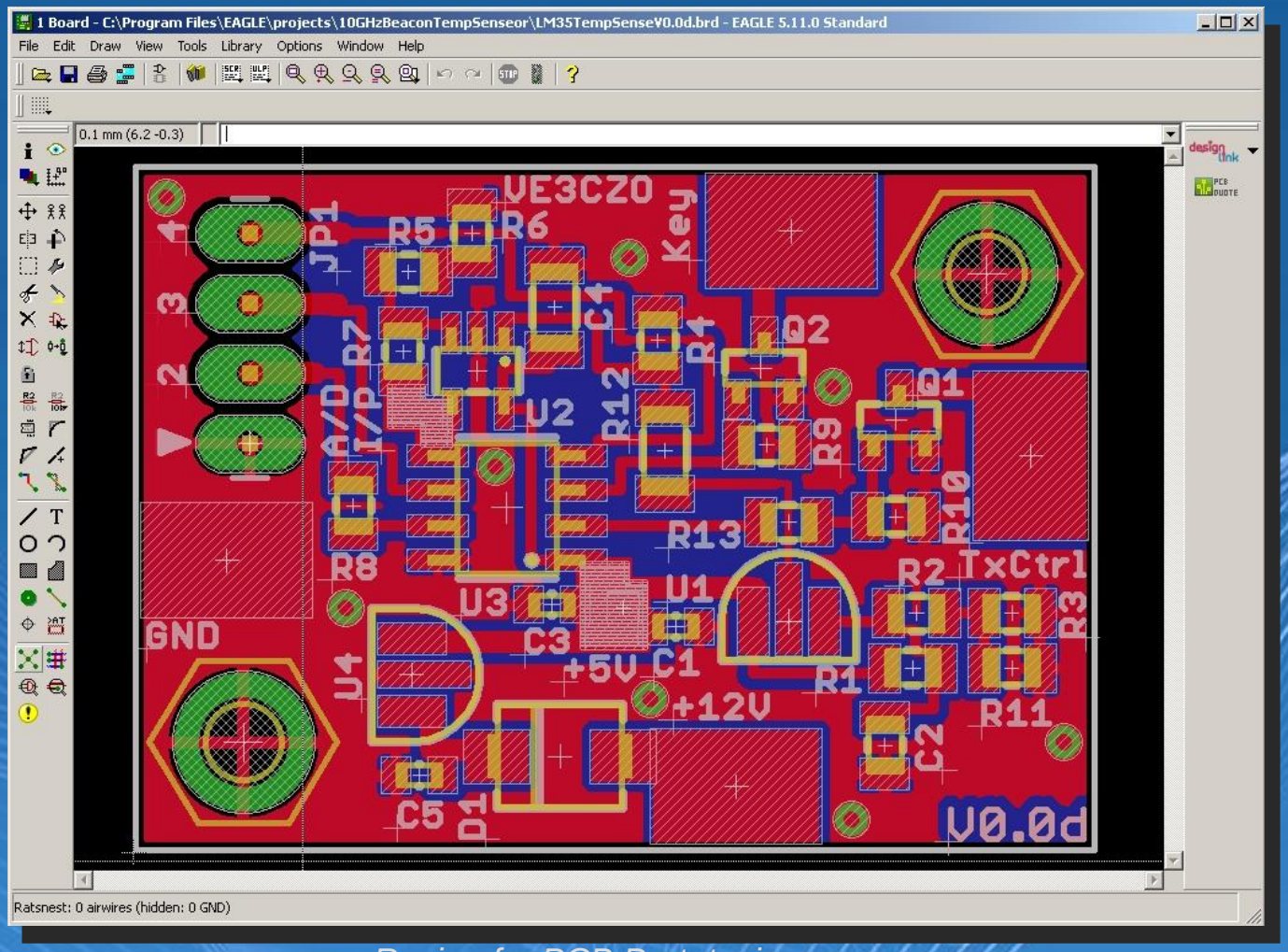
And a little more of a challenge to route



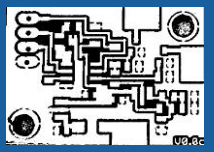
PCB Layout



Tracks routed & ground plane added

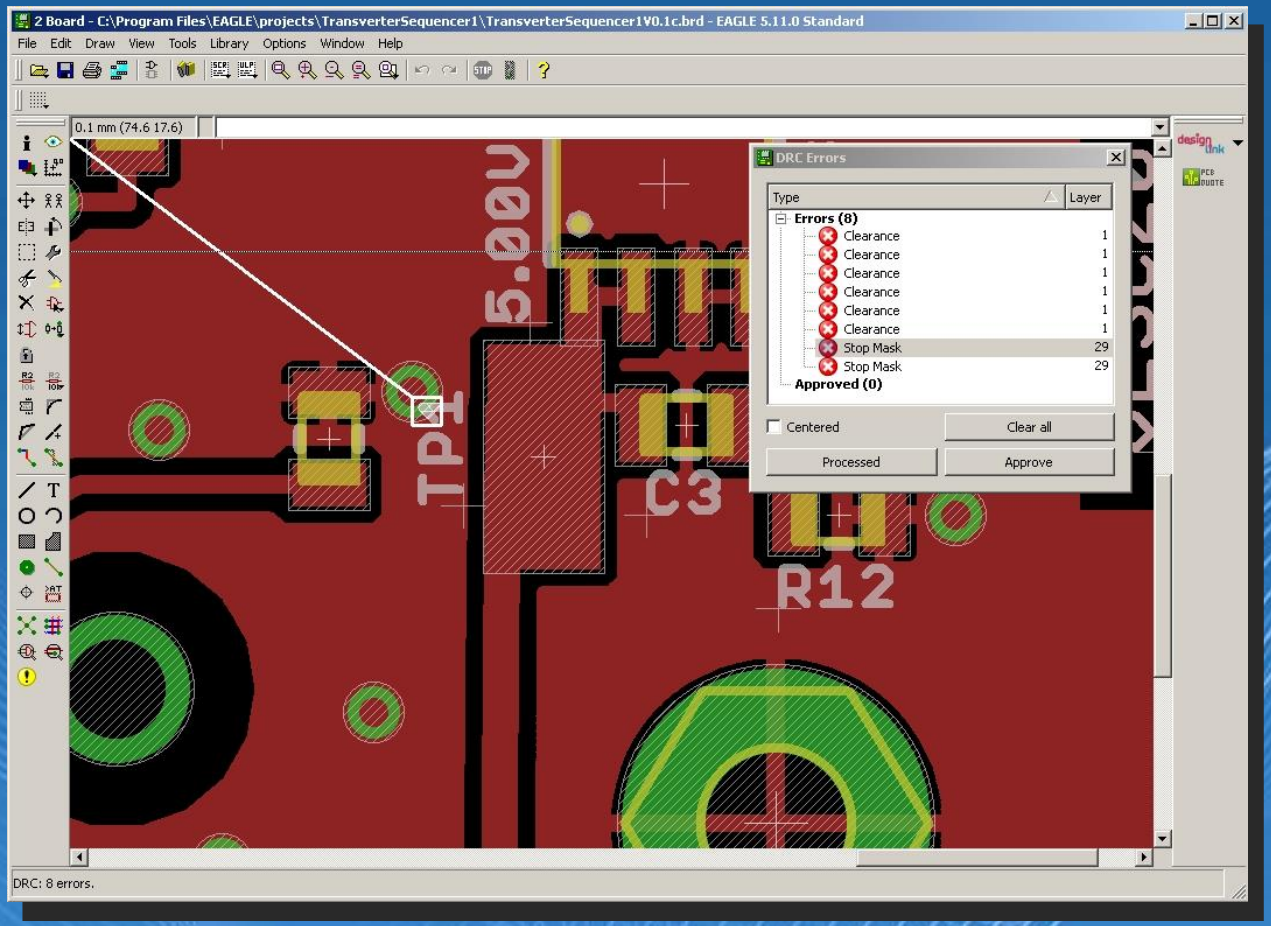


ERC Check

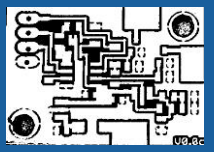


Run the ERC check with all needed Gerber layers active

- Top & bottom metal, solder stop masks & silk screen masks



Gerber tooling generation



Layers are automatically created

- Eagle processes PCB tooling files using a 'CAM' job

The seven Gerber files required for a 2 layer PCB

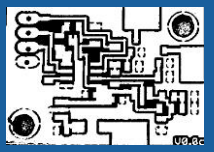
- GTL - top metal layer
- GBL - bottom metal layer
- GTS - top solder stop mask
- GBS - bottom solder stop mask
- GTO - top silk screen mask
- GBO - bottom silk screen mask
- TXT - NC drill file

Examine each layer carefully before committing to fab

- Small things almost invisible in Eagle 'pop out' when the Gerber files are viewed individually

CG Preview can be used to manipulate and view individual Gerber layers … and it's free!

Fab house or home process?



Two excellent low cost fab houses

- Both provide two layer PCB's with solder mask and silk screen
- Iteadstudio.com – 10 boards up to 5*5cm <$10 - up to10*10cm <$25
- batchpcb.com - $10 setup fee + $2.50 /sq in.
- Turn around time is 4 to 5 weeks

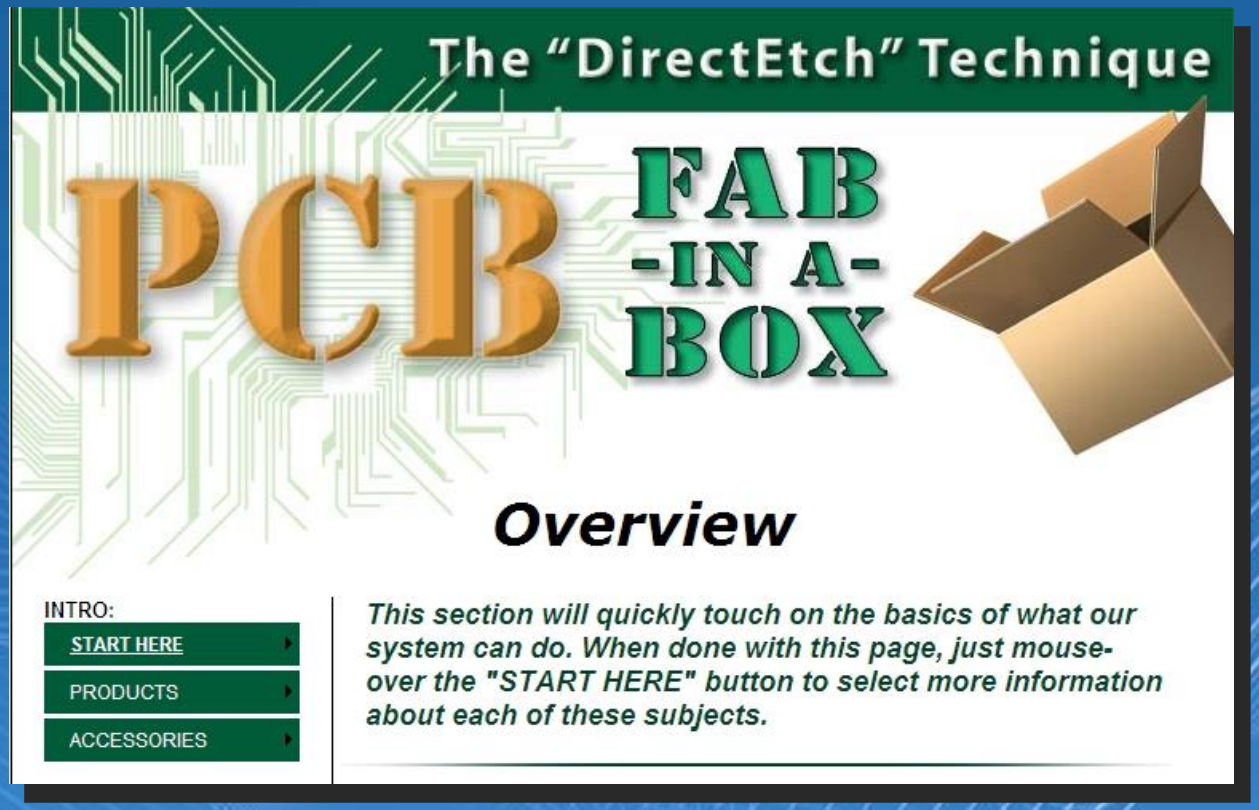
So why homebrew PCB's?

- Speed – boards are ready 2 to 4 hours after the artwork is ready
- Prototype before committing to PCB service especially RF stripline and coplanar waveguide designs
- Low cost <$0.30 /sq in once process equipment in place
- And some negatives
  - No plated through holes
  - Line and spacing pitch is not as fine as commercial PCB fab
  - Initial equipment cost consumes the savings for first ten or more batches
  - Process prove-in trial and error can time consuming & frustrating

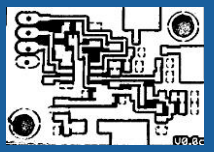
Homebrew process recipe

Process based on PCB Fab-in a-Box

- www.pcbfx.com – good tutorials and plenty of good advice
- Direct transfer from laser printout on transfer medium to PCB



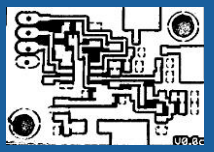
PCB Fab-in a-Box



Feature sizes usable with this process

- Minimum line size and spacing depends on laser printer resolution & print quality
- Line width and spacing - 0.25mm (10mil) min, 0.4mm typical
  - Commercial fab houses typically hold to 0.15mm (6mil) min
- Vias - 0.66mm insure through board connectivity with reasonable alignment care
  - Commercial fab houses typically allow minimum 0.6mm vias with 0.3mm hole

Fab-in a-Box Components



B&W laser printer

1200 dpi resolution
Straight through paper print path
Cardstock paper thickness
Toner density selectable

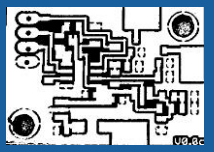
Toner Transfer paper

Sealer – Toner Reactive Foil

Toner Image Applicator

- Pouch Laminator GBC model H-220

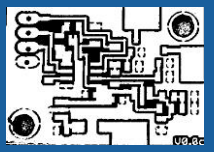
Preparing Artwork…
Tiling PCB's



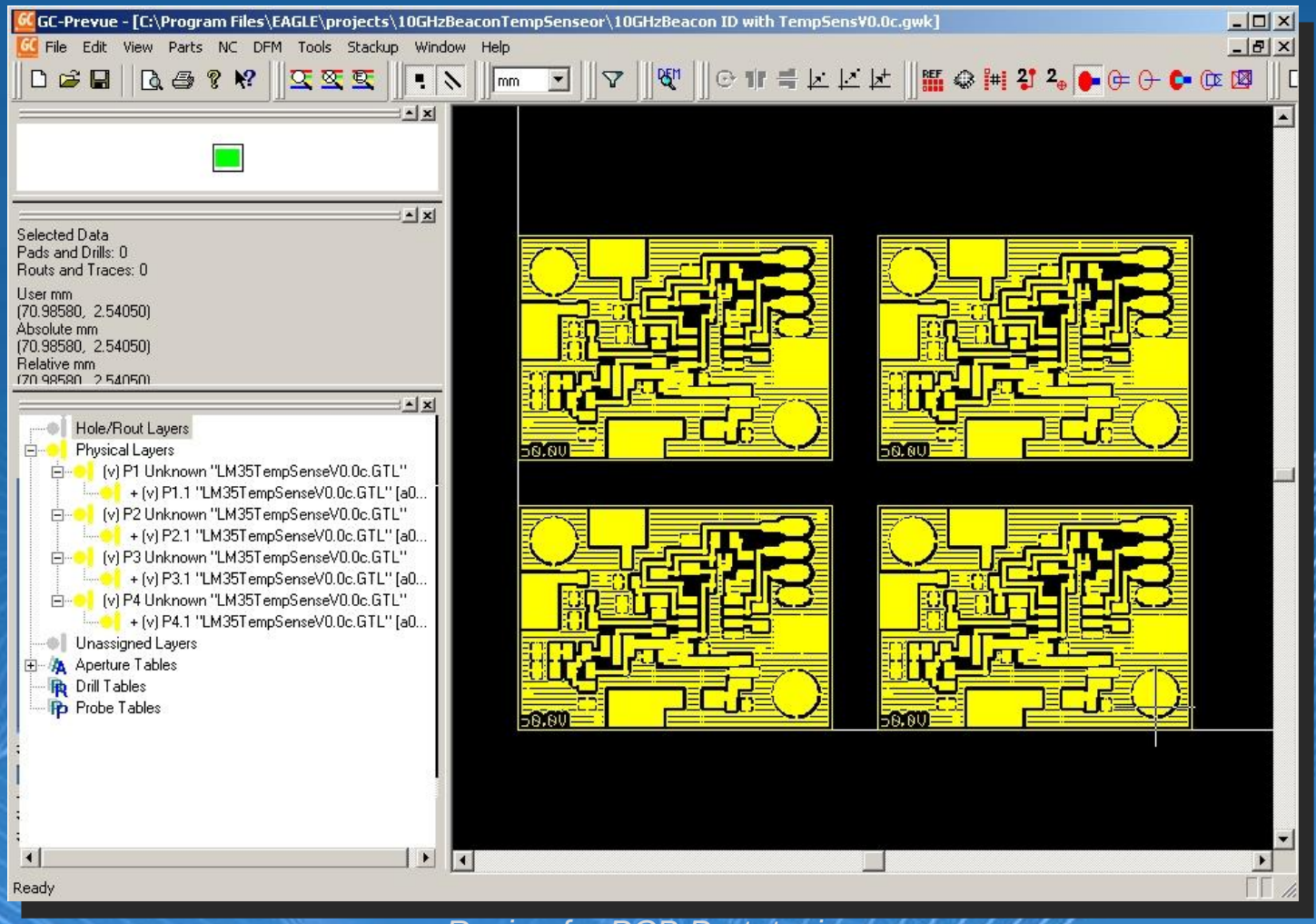
Tile top and bottom copper layers (GTL & GBL) using GC Preview

- Boards less than 5cm (2") on a side won't run through the laminator so need to be tiled. It's also often useful to run multiple copies of a design or multiple boards at once so tiling the PCB design and adding alignment fiducials for double sided board is the next step
- Detailed procedure in available documentation
- Briefly orient components as follows
  - Mirror top layers but not bottom layers
  - To minimize the amount of copper to be etched Leave 5mm between tiles in a cluster, 20mm between top and bottom layers
  - Align most critical spacing so that it's normal to the printer paper direction to minimize printer toner scatter errors

GC Preview

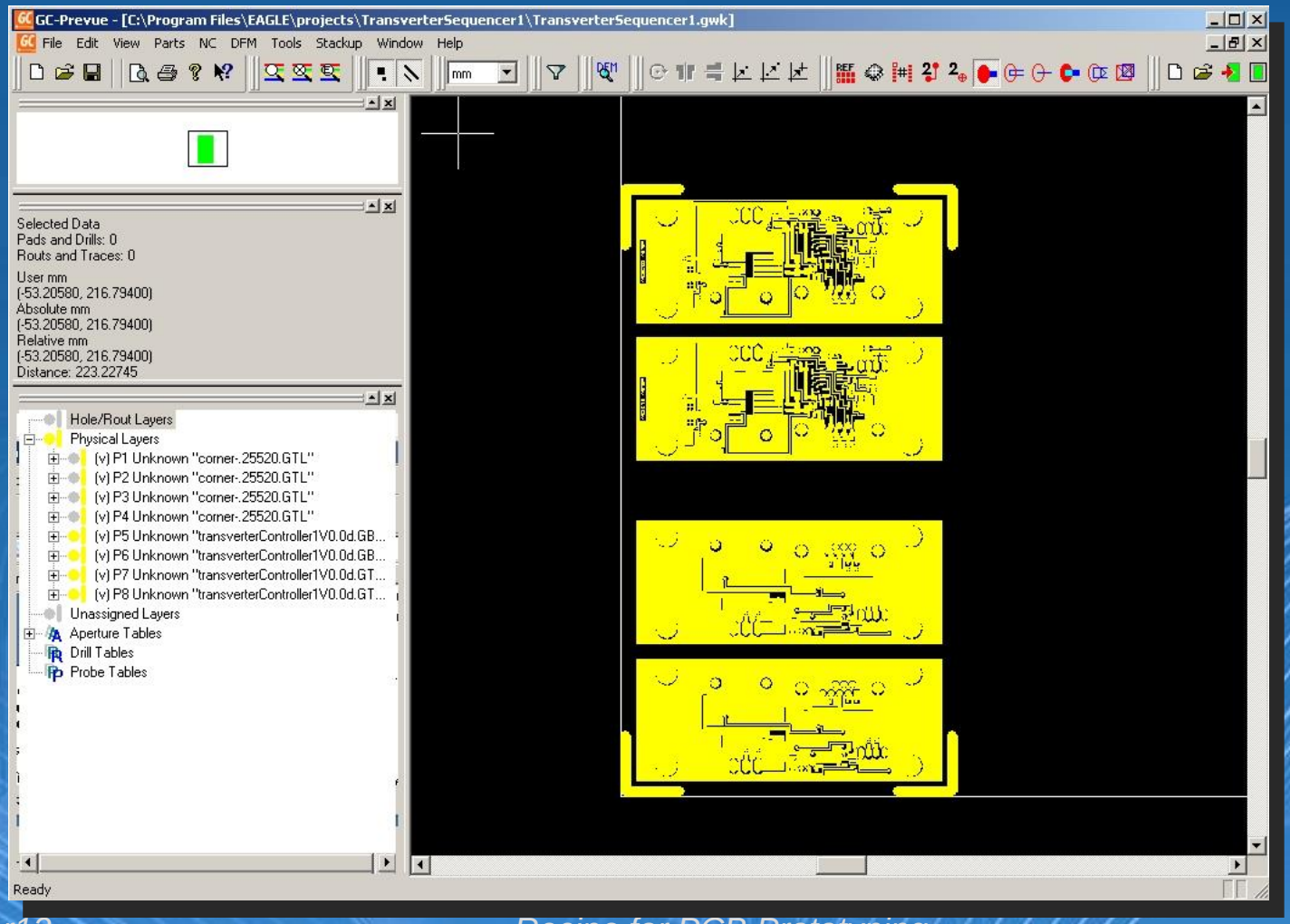


Tile for single layer PCB

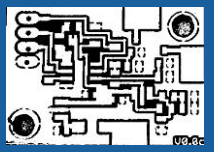


GC Preview

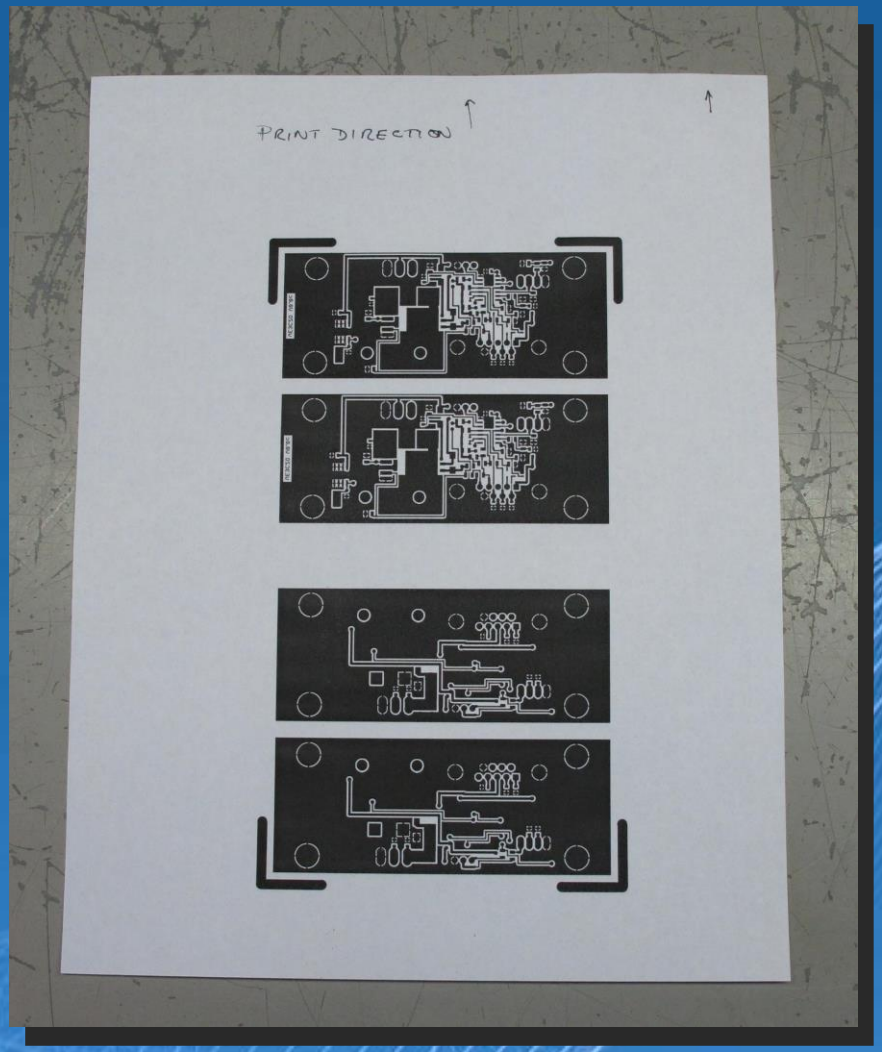
Double sided board tiled with alignment fiducials



Print PCB on std. paper



GC Preview print out of tiled PCB array centered on standard 8.5 x 11” paper

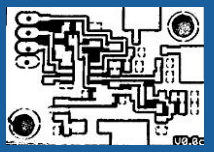


Humidify Transfer Paper

Place Toner Transfer Paper over water bath

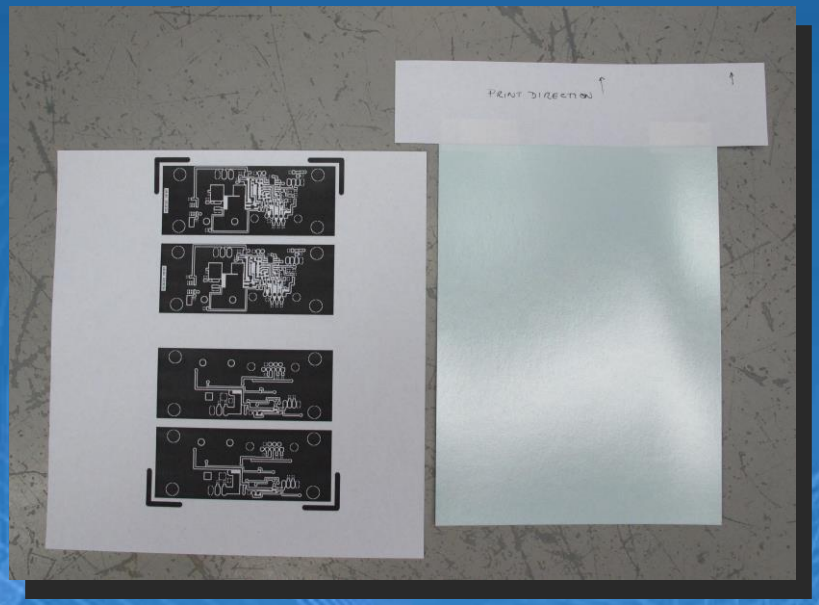
- Cut to size based on GC printout
- Let rest 2 minutes over bath – improves toner adherence to transfer paper

Print on Transfer Paper

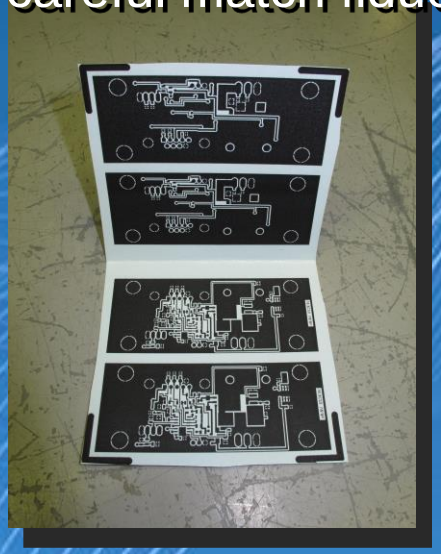


Prepare transfer paper

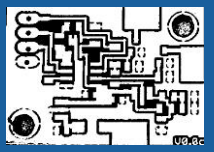
- Cut the standard paper print just above the top alignment fiducials
- Attach Transfer paper to the cut header using Scotch Removable tape
- Set printer for cardstock and highest toner density



- Cut tiled transfer paper to fiducial alignment marks and fold to careful match fiducials



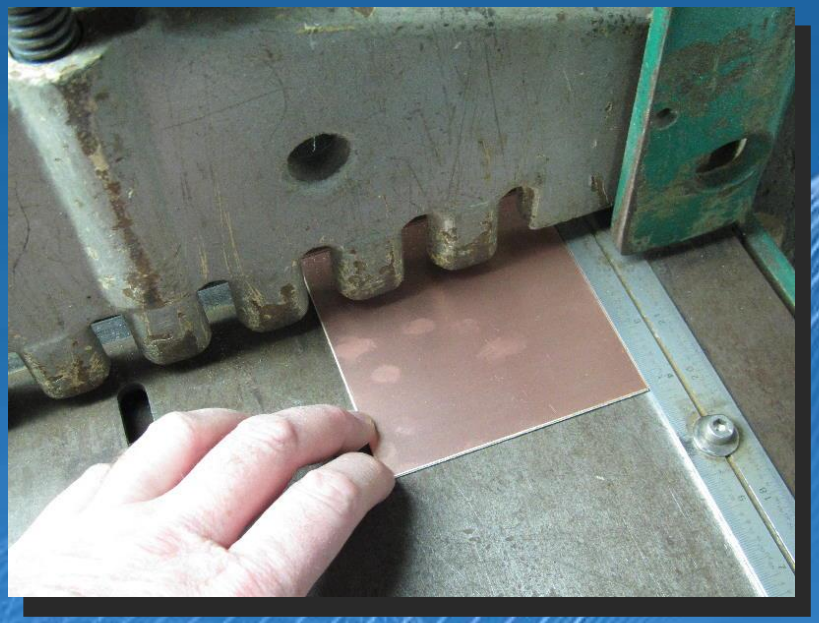
PCB Blank



PCB material

- FR4 1.6mm (62mils) with 35u (1oz/sq.ft.) copper is standard and low cost
- Other FR4 options – 32mils 1 or ½ oz also good
- Stick with FR4 especially for RF boards

Use a shear, hacksaw, coping saw, or Dremel tool with cut-off blade to cut the PCB to size



Clean the blank PCB

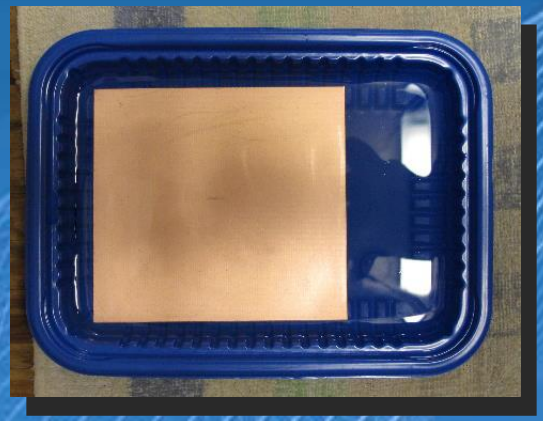
Start with household cleanser

- Remove tarnish fingerprints and any dirt

Finish with vinegar and salt bath

- 1 tablespoon salt in 1 cup vinegar heated to about 60 deg. C.

Rinse thoroughly and pat dry with a lint free shop towel to avoid water marks

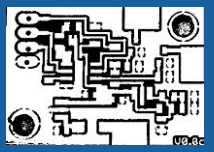


Laminator

Transfer paper folded over onto PCB is passed through the laminator

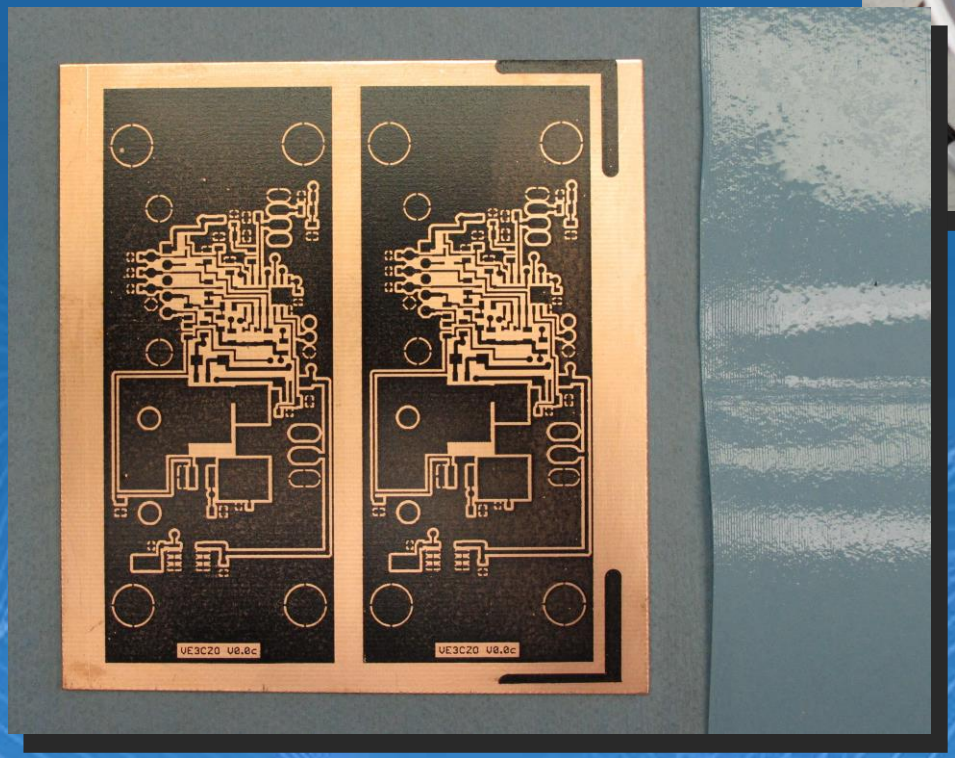
- Pass through, fold first. Flip over and pass through again
- Laminator provides correct heat and pressure optimizing toner transfer onto PCB

Quenching Transfer Paper



Bucket filled with water

- Takes about 30 seconds for the transfer paper to release toner onto PCB



TRF foil

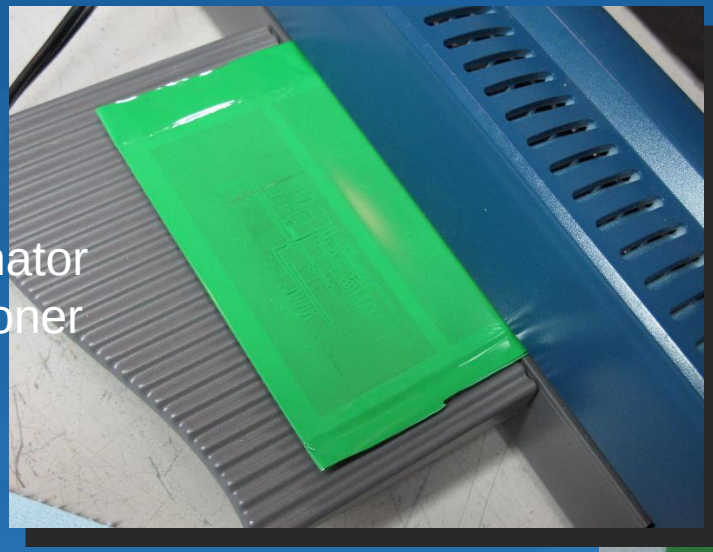
Toner Reactive Foil

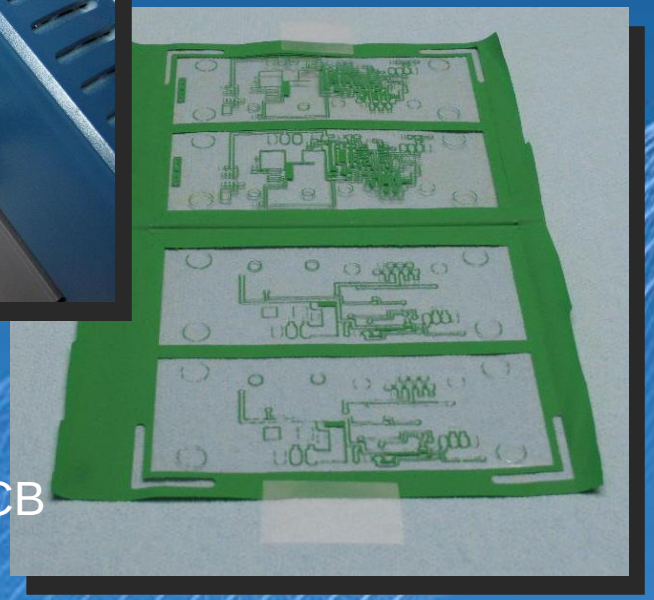
- Seals laser printer toner deposited on copper
- Pass through laminator fold over edge first. Flip PCB and pass through laminator a second time

TRF foil

TRF foil is deposited onto the PCB

Foil exiting laminator impressed into toner



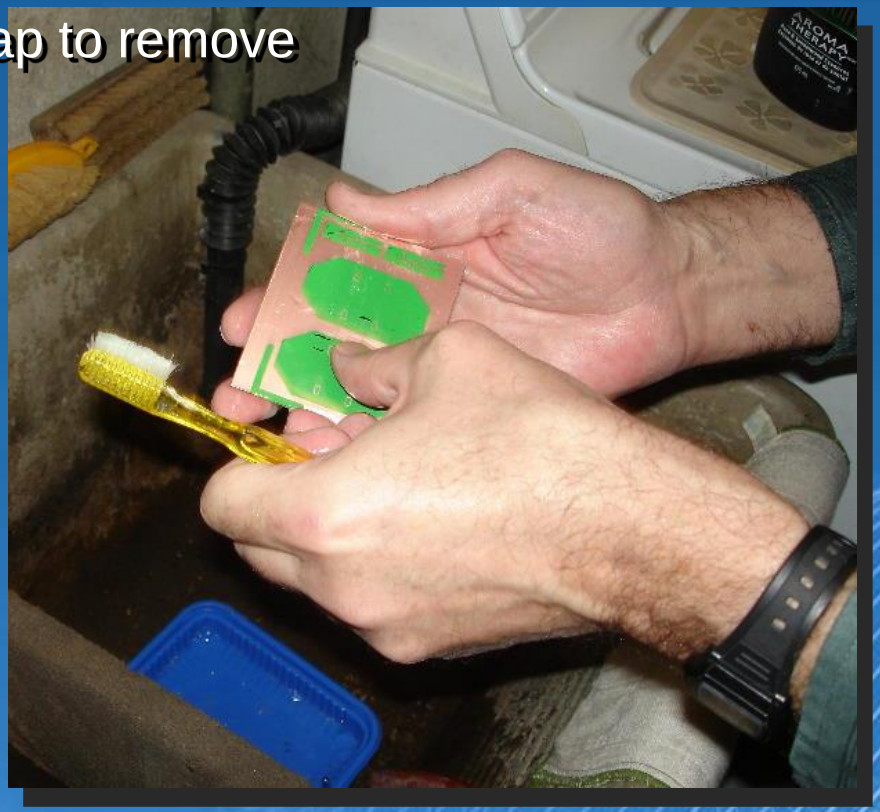


Spent TRF showing areas transferred to PCB

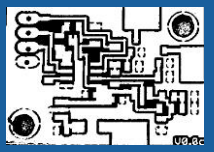
Cleaning up TRF residue

Excess bits of TRF foil may be left on PCB

- Brush with a toothbrush and liquid soap to remove loose TRF

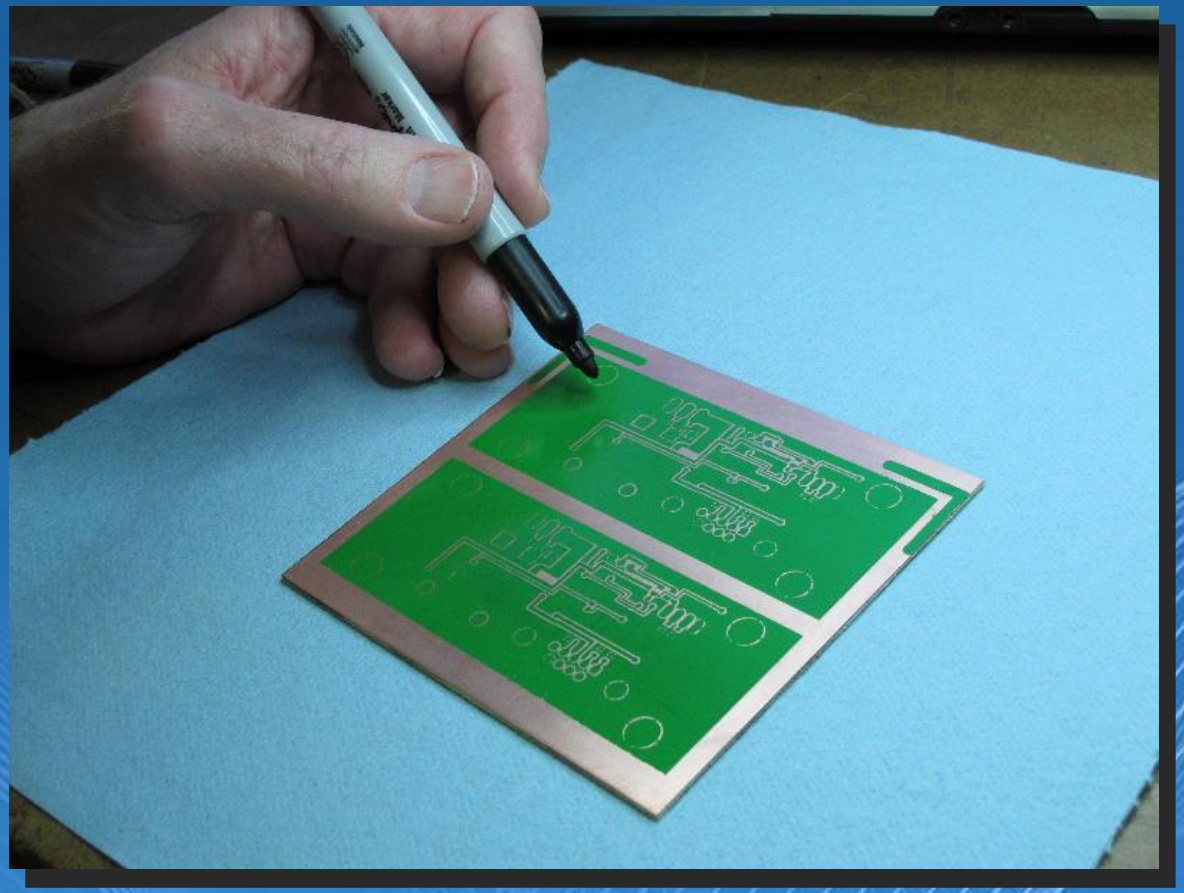


Cleaning Up Errors



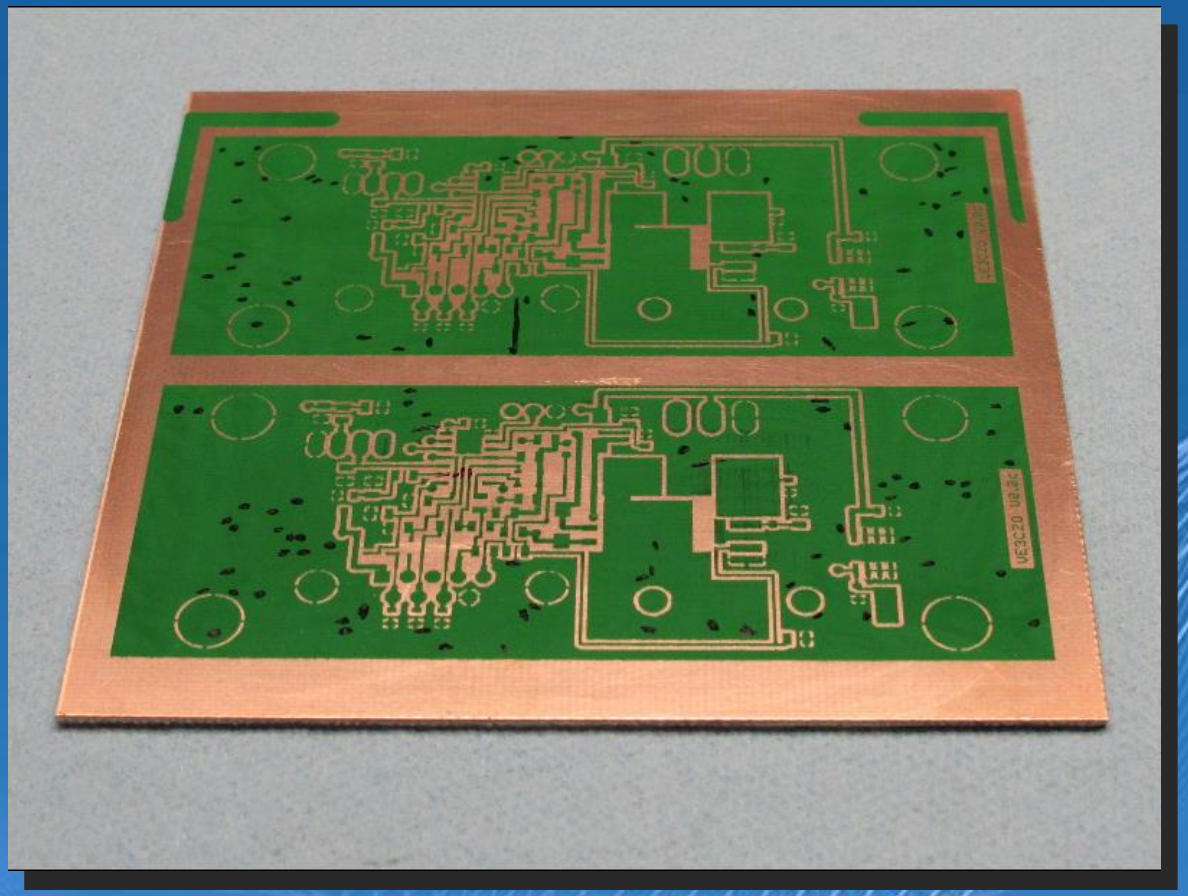
Sharpie marker fixes voids

- Inspect board very carefully under magnifying lamp



Preparing PCB for Etch

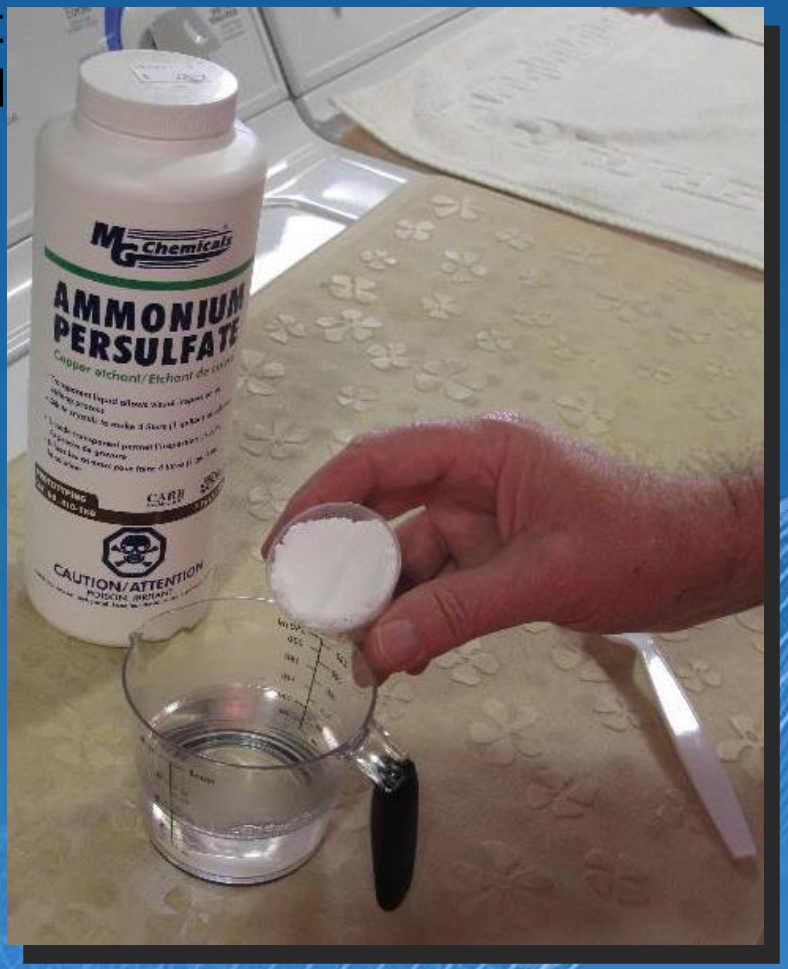
Sharpie ‘cleanup’



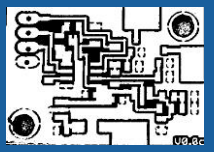
Prepare Etchant

Ammonium Persulfate

- Mix 4 parts water to1 part persulfate if new 3:1 if old
- Heat to 70 - 80 deg C

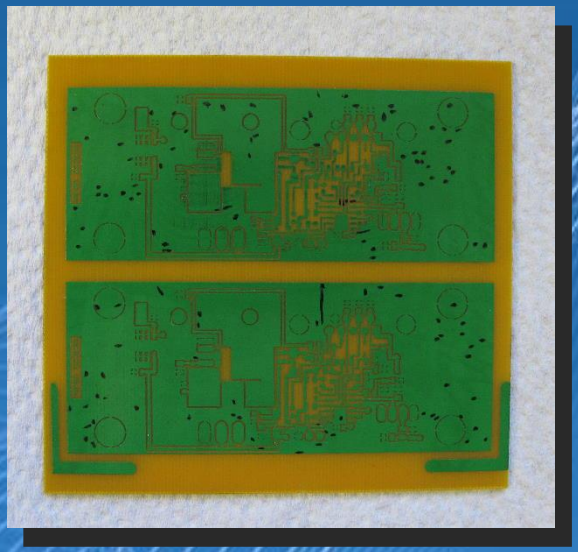


Etched PCB



Place etchant and PCB in a Ziploc plastic bag

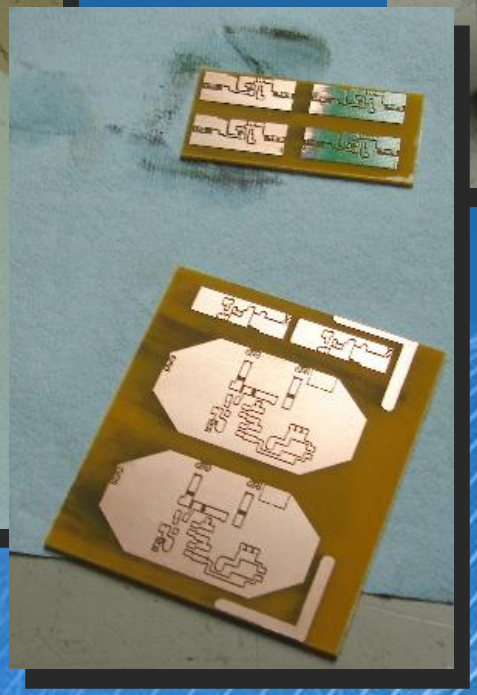
- Double bag if spillage is a concern
- Use a brayer and gently roll it over the PCB
- Turn the bag over occasionally and continue moving the etchant over the PCB surface with the brayer



- Etch time approx 10 min.

Clean the Etched PCB

- Remove TRF and toner using Acetone soaked onto a paper towel
- Clean the etched PCB using the salt & vinegar bath used initially to clean the board then rinse thoroughly



Electroless Tin Plating

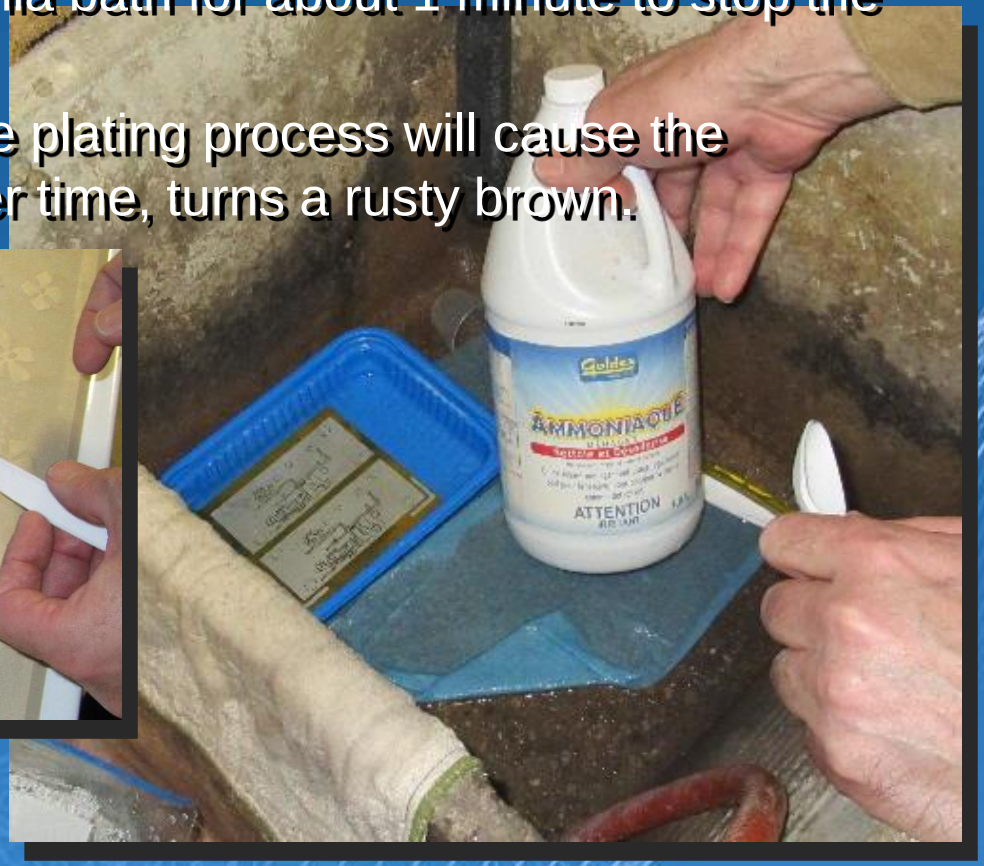
Tinnit

- Two part dry mix 5cc each part (teaspoon) in 70cc distilled water
- Indefinite shelf life in dry form, 2 months when mixed
- Heat water to 60 – 65 deg C and add plating salts
- Stir until salts dissolve, keep heating the bath or salts will begin to precipitate out
- The PCB must be clean!
- Immerse PCB in hot plating bath for about two minutes

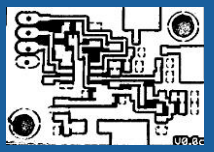
Tin Plating

Neutralize plating reaction

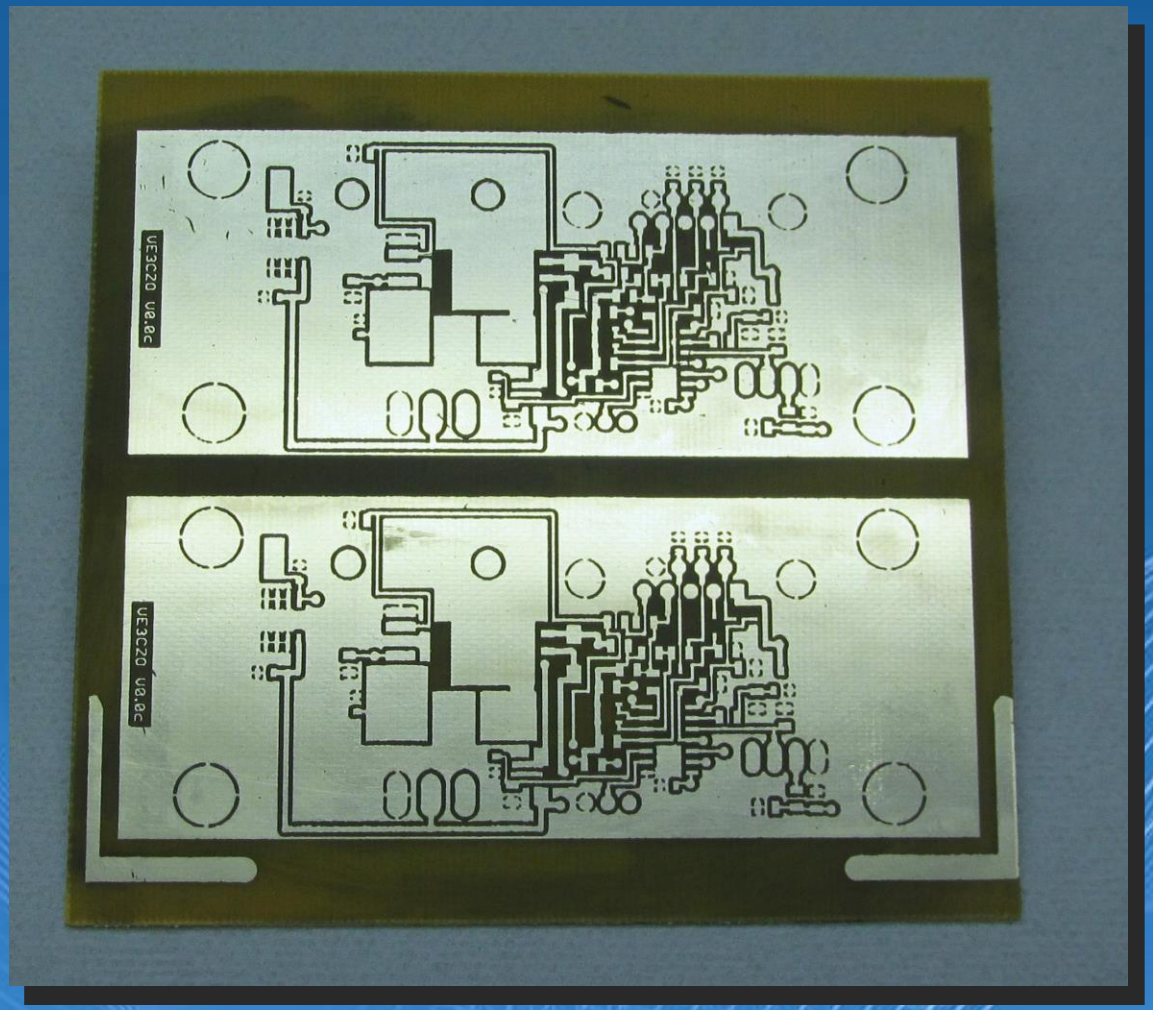
- After sufficient plating has been deposited on the PCB dip the board in an Ammonia bath for about 1 minute to stop the plating process.
- Failure to neutralize the plating process will cause the plating to discolour over time, turns a rusty brown.



Etched & Plated PCB

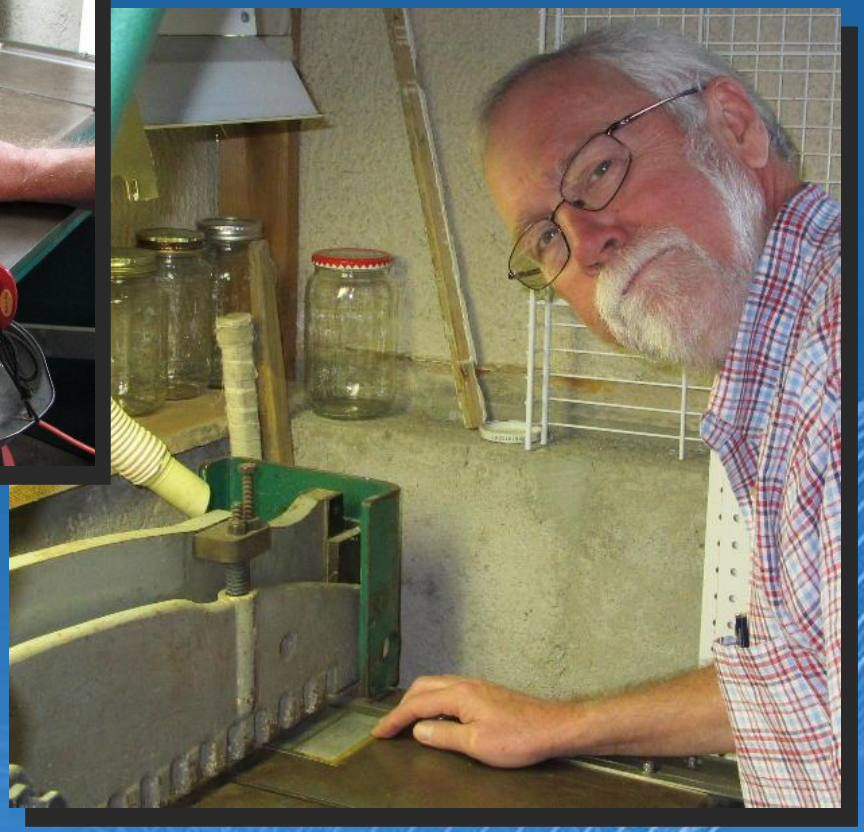


Completed etched and plated PCB still in tiled form



Separate Tiled PCB's

Use a Shear, hacksaw, coping saw, or Dremel tool with a cut-off blade – or ask Bryan

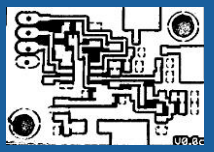


Finish PCB Trim

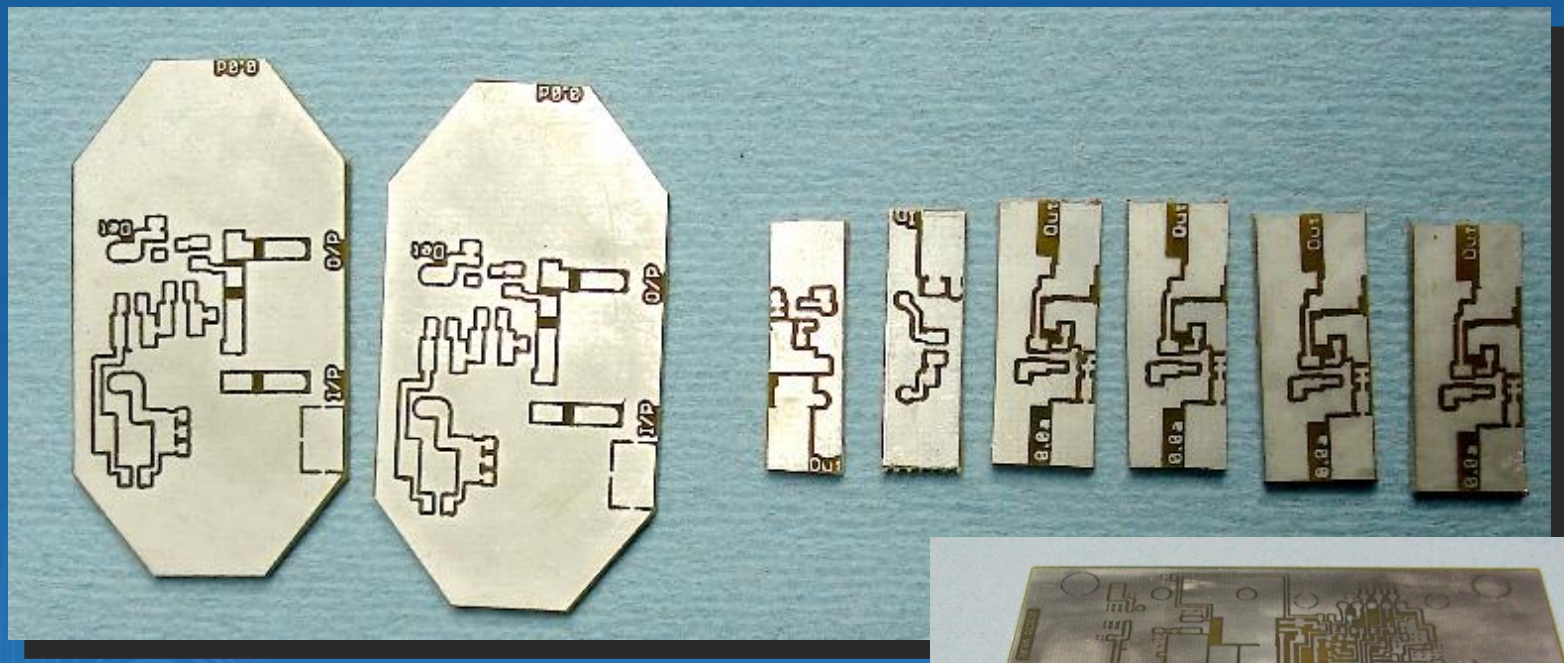
Disk, belt sander, or file

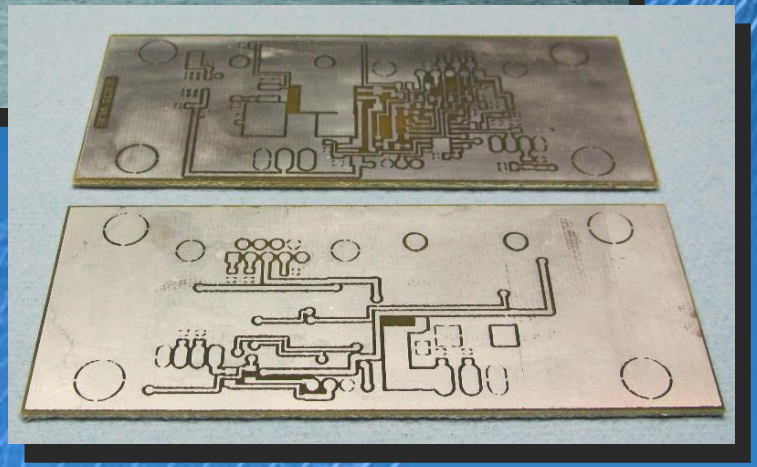
- The inexpensive Craftex disk / belt sander is ideal for this task

Etched Plated PCB



PCB's trimmed and sanded to final size



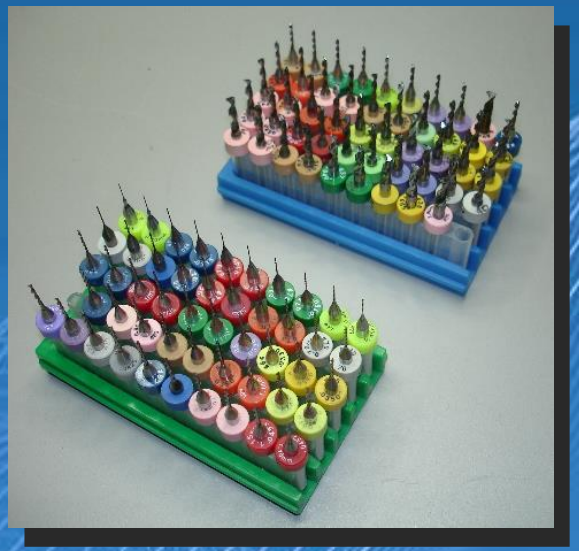


PCB Drilling tools

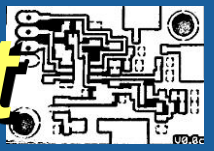
Proxxon mini-drill press

A task light is essential - Ikea Jansjo LED desk lamp works well

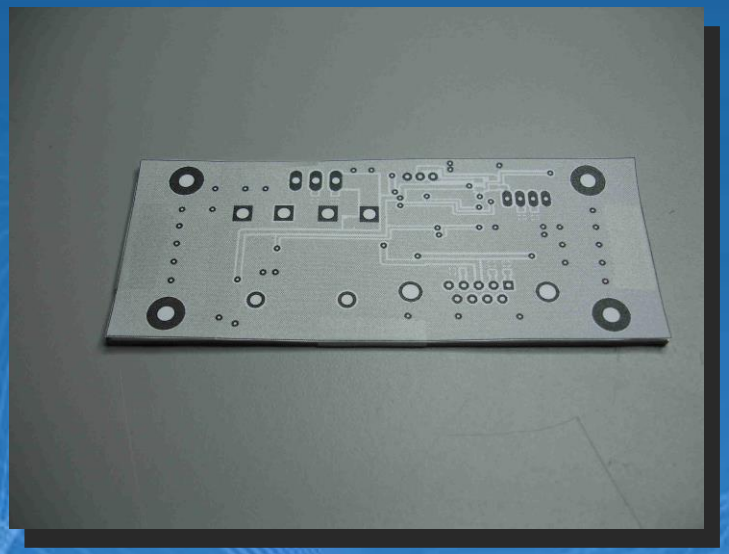
PCB drill bit sets available from eBay

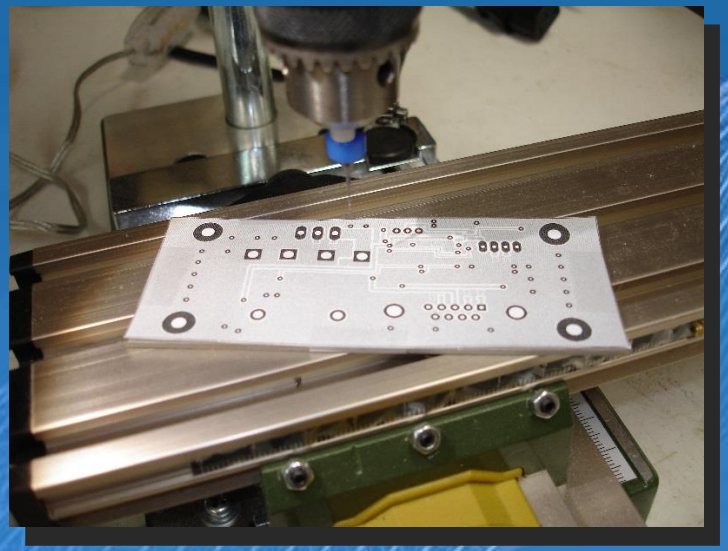


Getting the PCB holes right

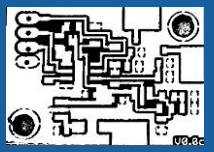


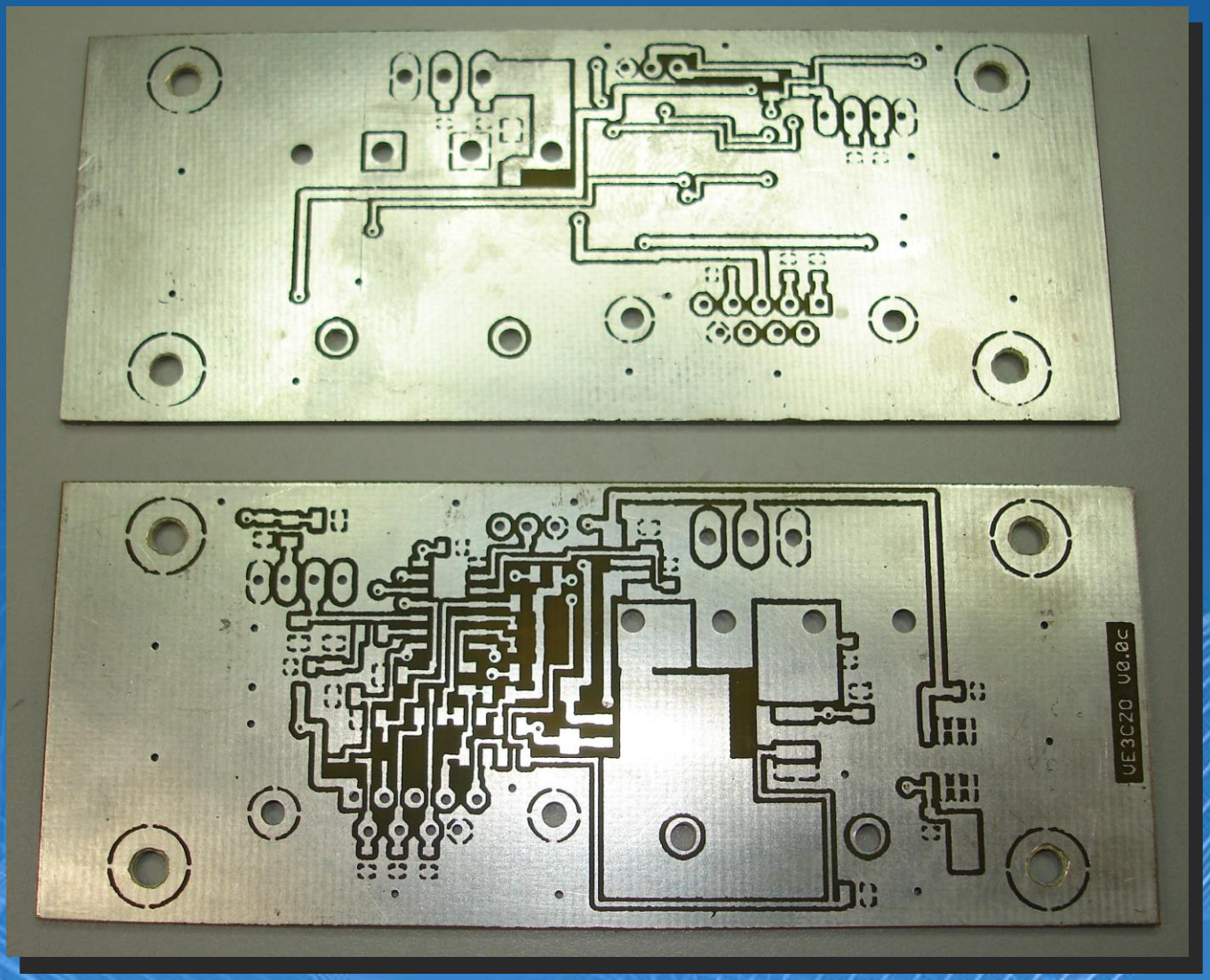
- Keep a list of drill sizes used for common holes – speeds up the drilling task
- Print out and carefully align a 1:1 drawing on the PCB and tape it in place to be used as a template.
- Don’t use the template method for non ground plane vias, drill them directly on the bare PCB for best alignment





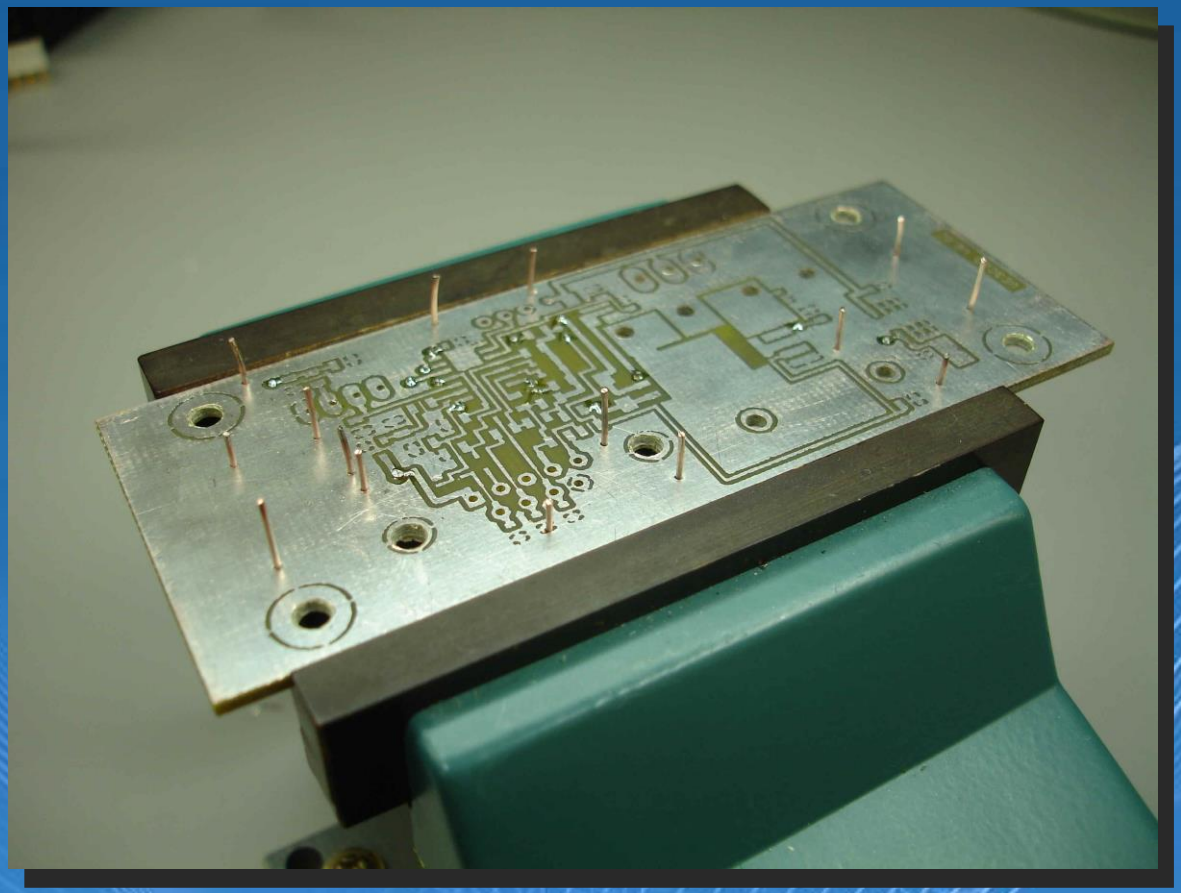
PCB Drilling Complete



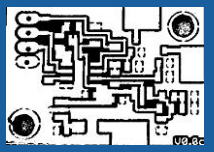


Drilling & Populating Vias

- use #76 drill 0.46mm & 30ga wire wrap wire
- RF ground plane vias use #70 drill 0.7mm & 22 ga wire. This wire gage makes the via fairly low inductance



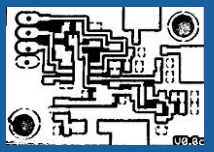
Equipment & Supplies



Capital Equipment & Vendors

- B&W Laser Printer with envelope feed through path
  - HP - Epson - Samsung etc.
- Pouch Laminator GBC H-220 www.dipmicro.com
- Proxxon drill press - TMB115 (+ chuck + XY table) www.minicrafttools.com or www.micromark.com
- Soft rubber brayer for print making www.michaels.com - Speedball (pressure roller)
- PCB drill sets - eBay search on 'PCB drill bit' - bitguy1
- Illuminated magnifying lamp 3 diopter - Busy Bee Machine Tools
- Craftex disk / belt sander - Busy Bee Machine Tools
- Microwave oven - Kijiji
- Infrared thermometer - Canadian Tire or eBay
- Shear, hacksaw, coping saw or Dremel tool with cutoff blade

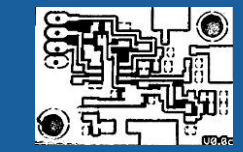
Equipment & Supplies



Consumable Supplies

- PCB-in a-Box transfer paper & foil www.dipmicro.com or www.digi-key.com
- Blank PCB - eBay search on 'FR4' or DipMicro
- Ammonium Persulfate - GC Electronics - Active Electronics
- Acetone
- Tinnit by Philmore-Datak www.abra-electronics.com
- Microwave resistant food trays (for immersion baths)
- Measuring cups, plastic spoons
- Distilled water
- Ammonia
- Ziploc Plastic bags - heavy duty
- Lint free shop towel - Scott shop towel roll - Canadian Tire
- Sharpie permanent marker - ultra fine point & fine point

Special thanks to…



Bryan Campbell VE3ZRK for suggestions and patience in developing the homebrew PCB fab process with me

Dave Green VE3TLY and Dave Conn VE3KL for the great camera work that helped bring pictures to this presentation