Building Ham Radio Electronic Projects

The Joy of Craftsmanship

Gary W. Johnson, NA6O

October, 2022

https://na6o.com

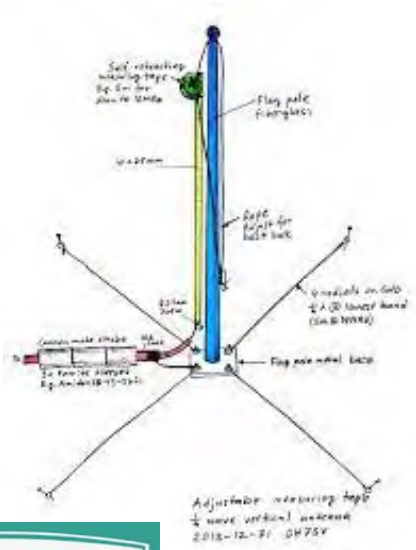
gwj@me.com

Hams have a long tradition of *homebrewing* equipment and it continues today

1940s Transmitter

Arduino Antenna Controller

Antennas, Antennas, Antennas.....



2000 Transceiver

In fact, many hams are almost 100% Makers rather than Operators!

HEATHKIT

Steps to building any project

- 1. Requirements: What does it need to do?
- 2. Design: Use or modify an existing design—most common way
- 3. Gather parts
- 4. Prototype: Build and test tricky parts first... Understand it!
- 5. Final build: Make it clean and robust, package it up

Even if you *could* buy it, why not enjoy making it yourself?

What tools do I need to make electronic stuff?

My “lifeboat” handtools

- **Needle nose pliers**
- **Diagonal cutters** Swanstrom
- **T-Strips**
- **X-acto knife**
- **Little vise, or some kind of clip to serve as a third hand**

- **Digital multimeter (DMM)**

Specialty tools for soldering

A good, temp-regulated soldering station

[Hakko FX951](#) $325

Nothing is more frustrating than a crappy soldering iron!

0.032 inch solder

Soldapult solder sucker

Tweezers for SMT parts

[Excelta Tools](#)

Solder wick

Magnifiers for SMT

[Mag Eyes](#)

Where do I get electronic parts?

- There used to be Radio Shack, Fry's Electronics, surplus stores, and others. Not so much anymore...
- Anchor Electronics in Santa Clara
- Some favorite sellers

  Jameco.com - In Belmont. Online, or an actual store!

  Electronix Express, elexp.com

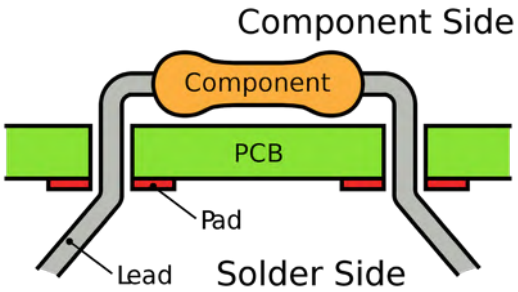
  Mouser.com - Often cheaper than DigiKey

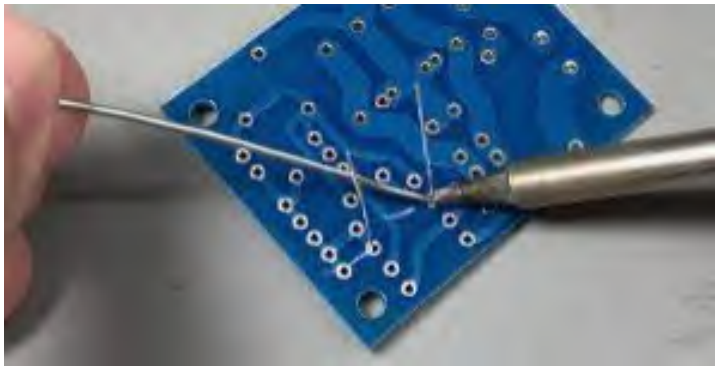
  DigiKey.com

  RFparts.com – New and surplus

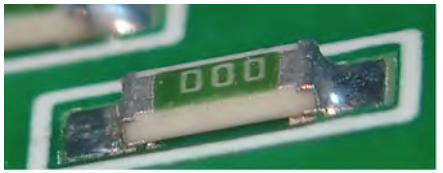
Through hole vs. surface-mount technology

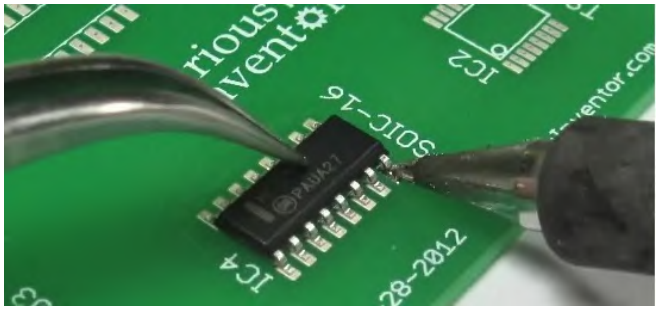
- **Through-hole is preferred for experiments and prototyping**
- **Bigger, easier to solder (and remove), easier to probe**
- **Not all ICs are available in this format**





- **SMT is more compact**
- **Much more challenging to solder**
- **Excellent for automated production**





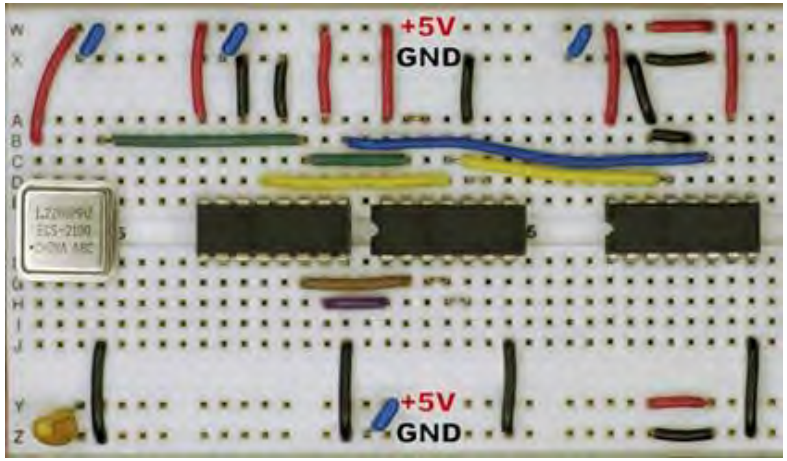
There are many ways to prototype a circuit, including fabricating a circuit board

- Solderless breadboards
- Pre-etched prototyping boards (*stripboards* or *perfboards*)
- “Ugly style” for quick, but high-performance prototypes
- Simple PC boards by hand–quick, easy, and really cheap
- Using PCB design software and commercial fabrication

Tradeoffs are speed, cost, and learning curve

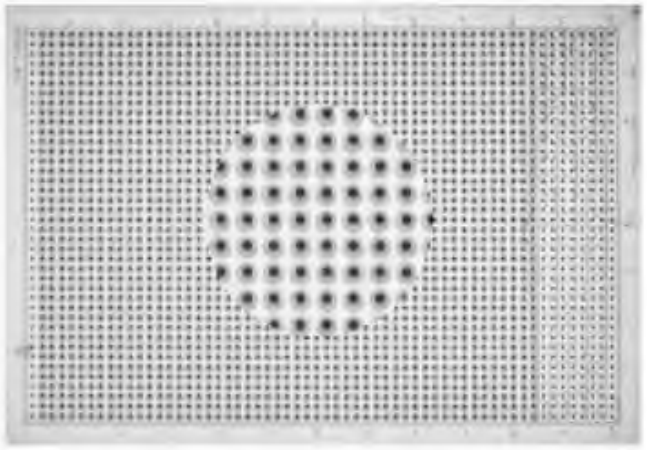
Solderless breadboards are best for quick tests, especially digital logic

- Very quick and easy to use
- Non-rugged, should never be deployed!
- Limited to low freq (< 1 MHz) due to uncontrolled capacitance and inductance



Perfboards give you a grid of holes with pre-etched traces, optional ground plane

- Cut to size and start soldering. 0.1-inch standard grid. Easy!
- Favorite model: Vector 8007, 4.5 x 6.5, ground plane. $28 @Mouser
- *Tip:* Put the ground plane on the component side



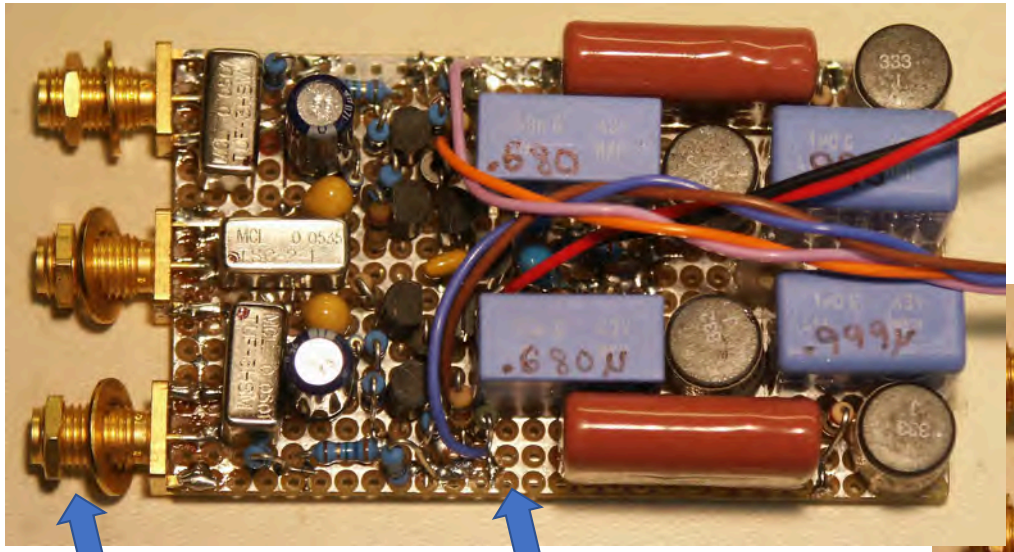
VECTOR

8007 **4.5" x 6.5"**

Circuit Pattern:	Pad-Per-Hole/ Ground Plane
Contacts:	N/A
Width/Thick:	6.50"/.062"
Height:	4.50"
16-Pin DIP Capacity:	60
Material:	CEM-1
Solder Terminals:	T42-1
Wire-Wrap Terminals:	T44, T46, T49, T68
Wire-Wrap Socket Pins:	R32
Hole Diameter:	.042"

- *Pad-Per-Hole pattern on component side - overall Ground Plane pattern on wiring side*
- *To commit wire-wrap pins to ground planes, use Vector T124 solder washers, available separately*
- *0.080" diameter, isolated solder pad around holes, component side*
- *Accommodates any type DIP IC device or discrete component*
- *Plane and pad surfaces solder-coated for user convenience*

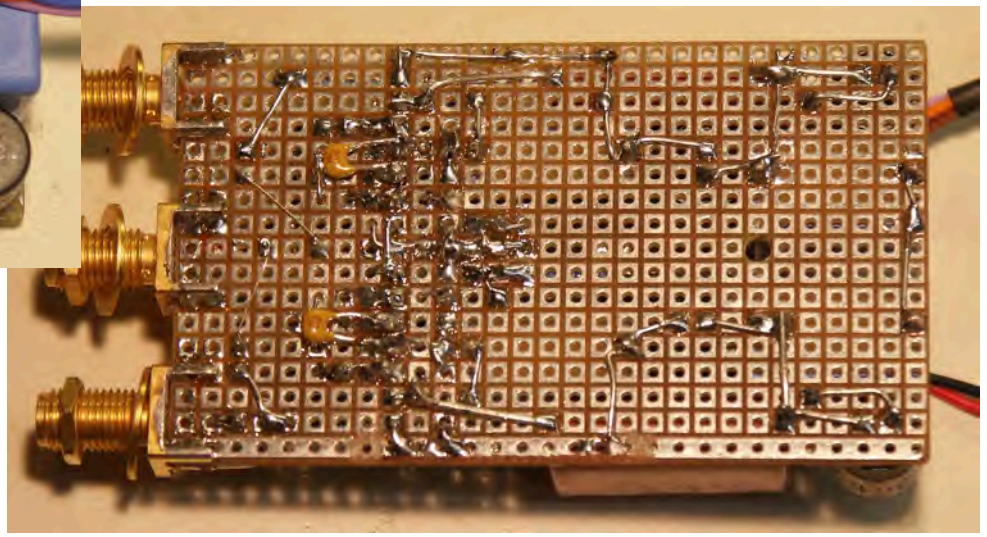
R2Pro direct-conversion receiver down-converter on Vector 8007 (Converts any radio frequency to audio)



Design by Rick Campbell, KK7B

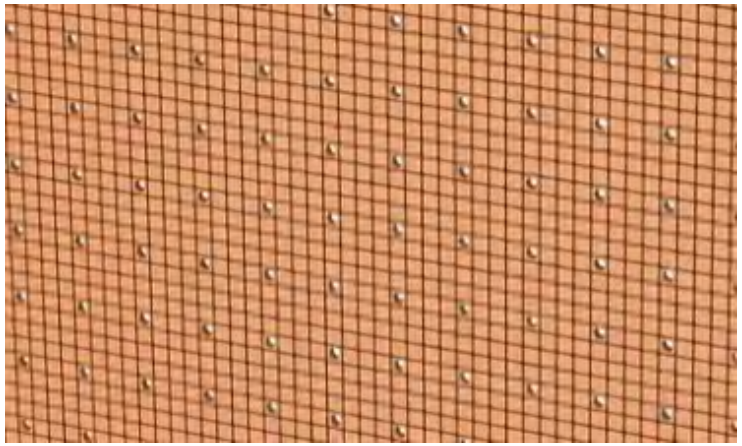
SMA connectors:
Fit nicely

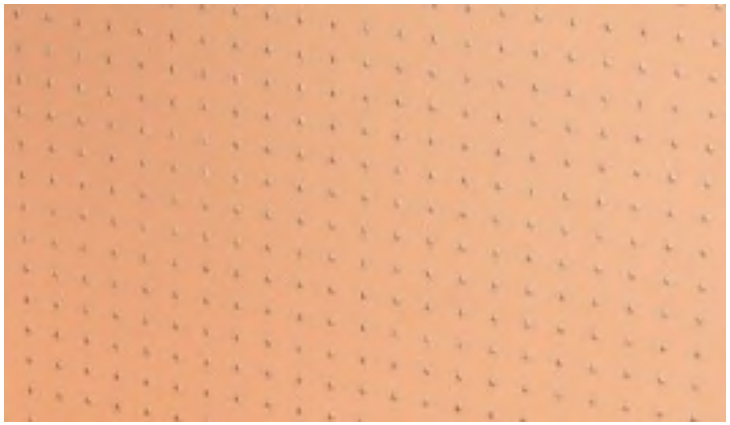
Ground plane
on component side



What about surface mount technology (SMT) prototyping?

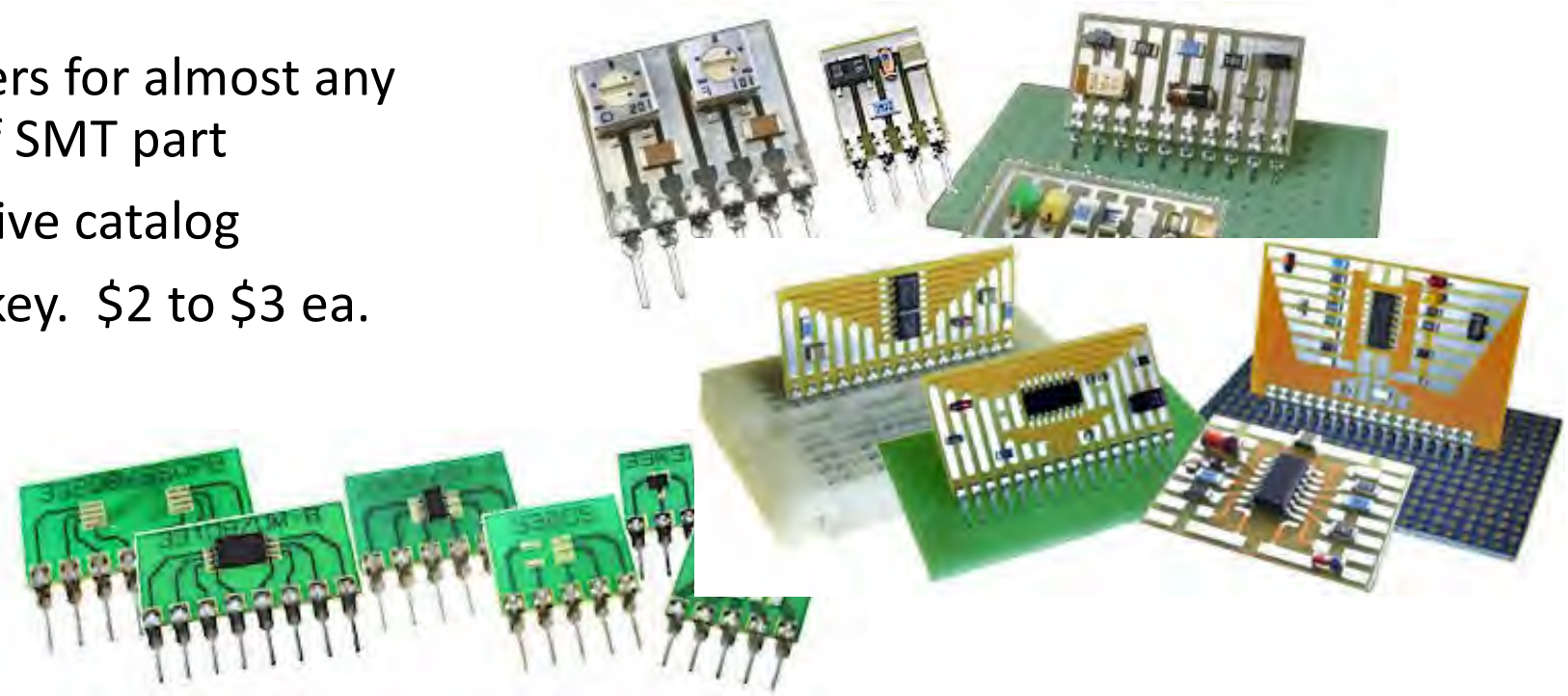
- **All Electronics SB-3T**, SMT proto board. $12.50 Link
- 100 x 160mm. Grid of SMT pads on 0.050" centers allows SMT components of various sizes and pitches to be mounted. Larger components can span pads. Solid copper ground plane on bottom of board. Every fourth SMT pad has an unplated hole for easy connection to the ground plane
- Ugly style also works well with SMT, but small pads require care...





Surfboards are also handy for SMT

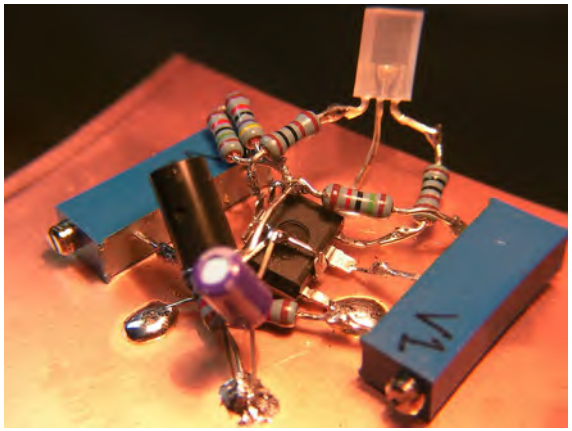
- Adapters for almost any kind of SMT part
- Extensive catalog
- @Digikey. $2 to $3 ea.



Capital Advanced Technologies

Ugly style prototyping is especially useful for RF, up through UHF

- Term coined by KK7B, co-author of *Experimental Methods in RF Design* (a FANTASTIC book, now out of print, sadly!)
- Based on bare copper clad board
  - Remember to clean the board first! Steel wool is great.
- Place ICs upside down... "Dead Bug" style
- Create pads with a Dremel tool with fine-point carbide tip
- Parts are tack-soldered anywhere you want
- Maintain substantial ground plane for RF work
- Can be very rugged, often deployed as QRP rigs and such

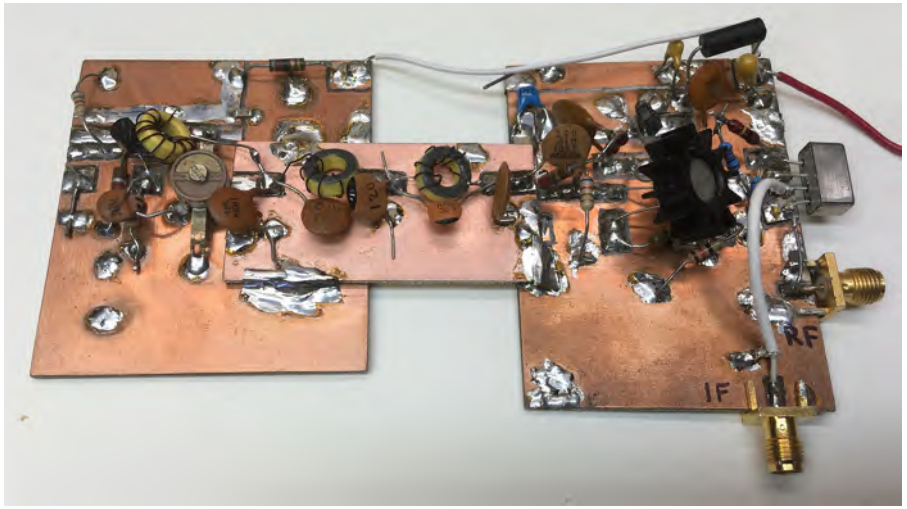


Some examples...

50 MHz low-noise amplifier

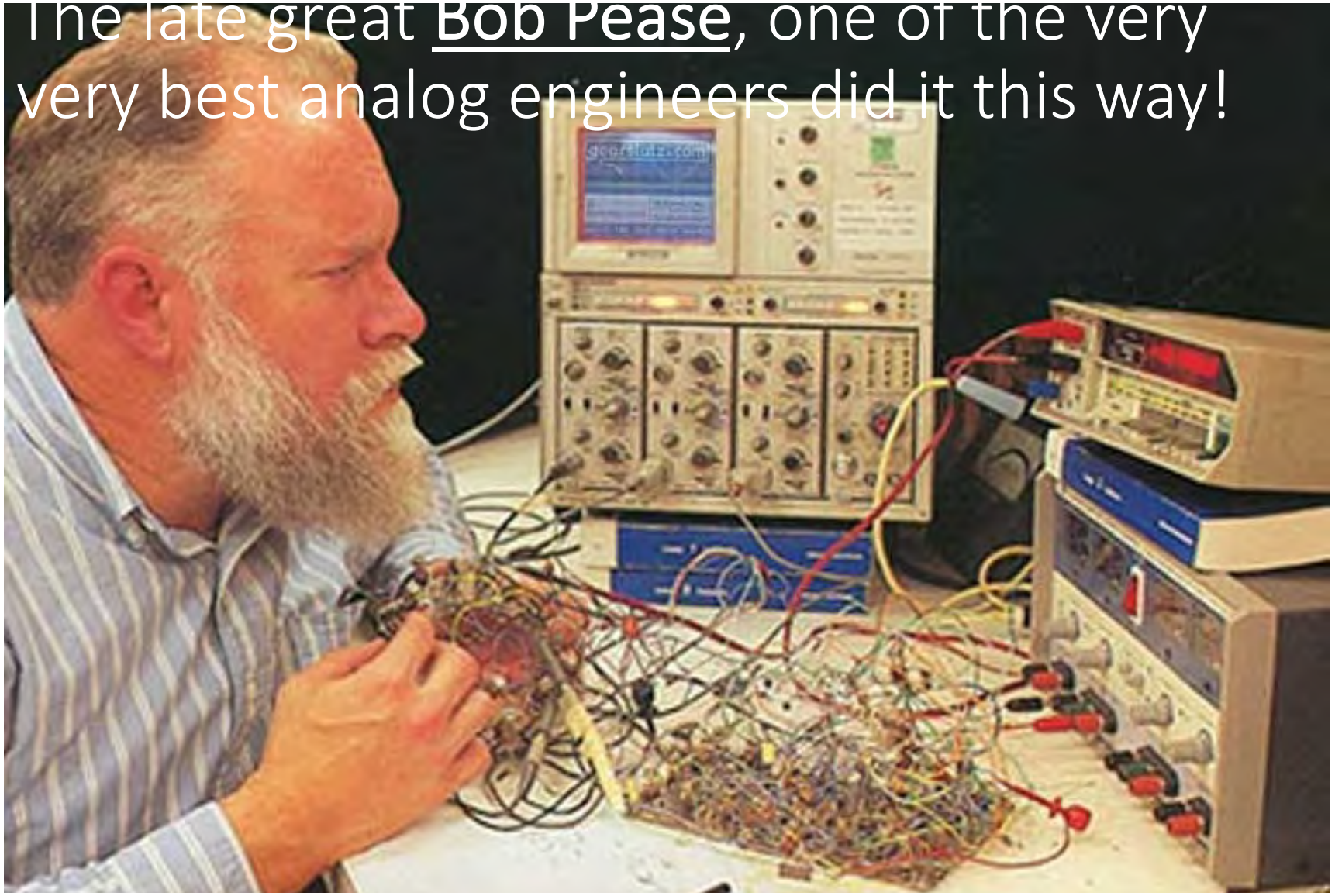
Tip: SMA connectors are great for these little boards

Receiver RF-IF chain built up from smaller modules. Build and test each one, then link together.



7 MHz QRP (low-power) transceiver by AA0ZZ

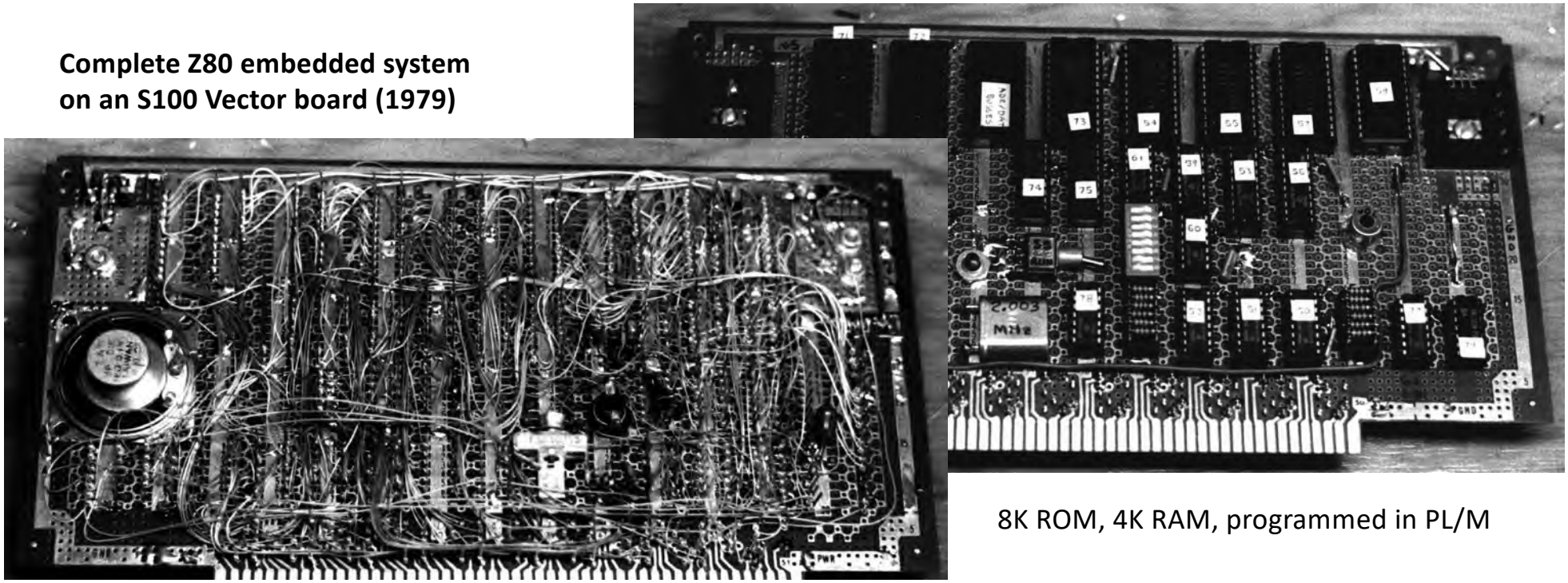
The late great Bob Pease, one of the very very best analog engineers did it this way!



Once upon a time... there was wire wrap!

- It's still around, sort of, and useful for digital ICs in particular
- Special wire, sockets, and tools required

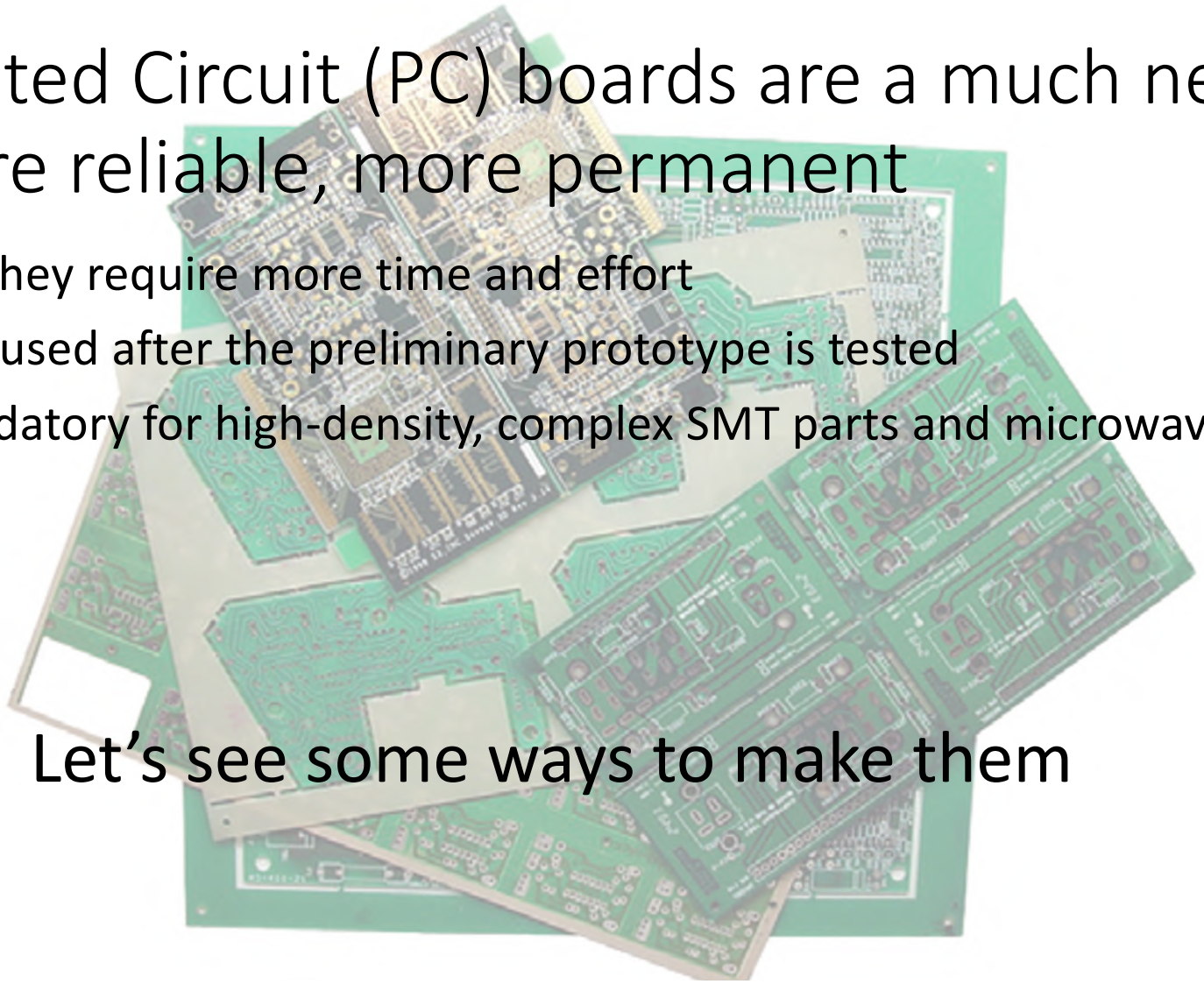
Complete Z80 embedded system on an S100 Vector board (1979)



8K ROM, 4K RAM, programmed in PL/M

Printed Circuit (PC) boards are a much neater, more reliable, more permanent

- But they require more time and effort
- Best used after the preliminary prototype is tested
- Mandatory for high-density, complex SMT parts and microwave

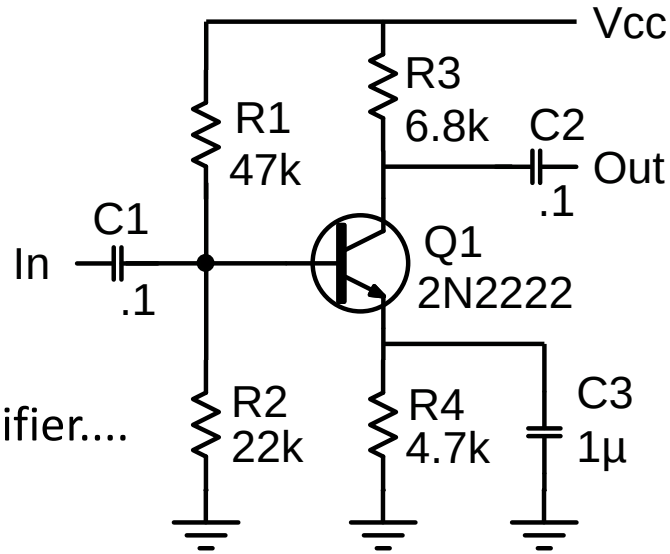


Let’s see some ways to make them

PC boards by the *Sharpie* method

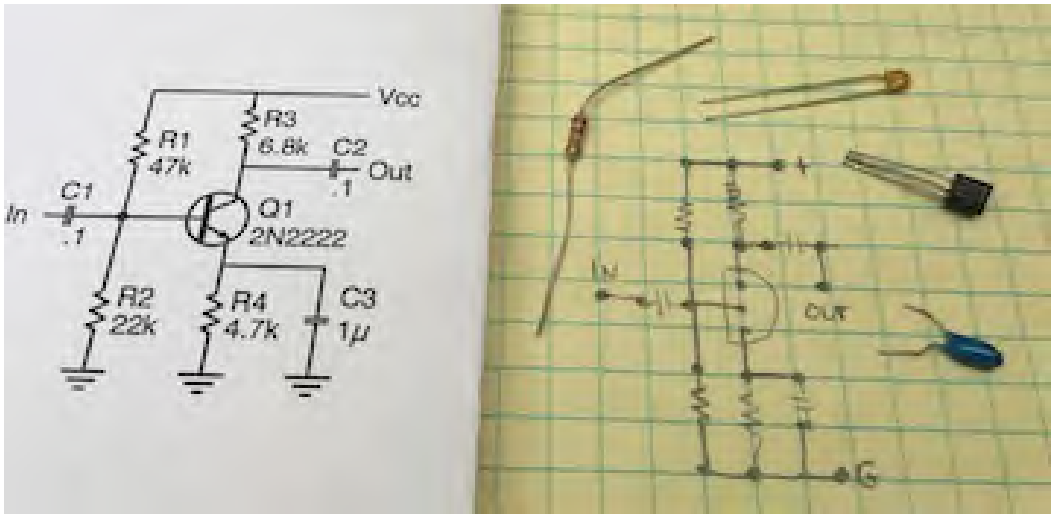
1. Draw your pattern on (clean) copper clad board
2. Etch
3. Drill holes

Let's built this simple amplifier....



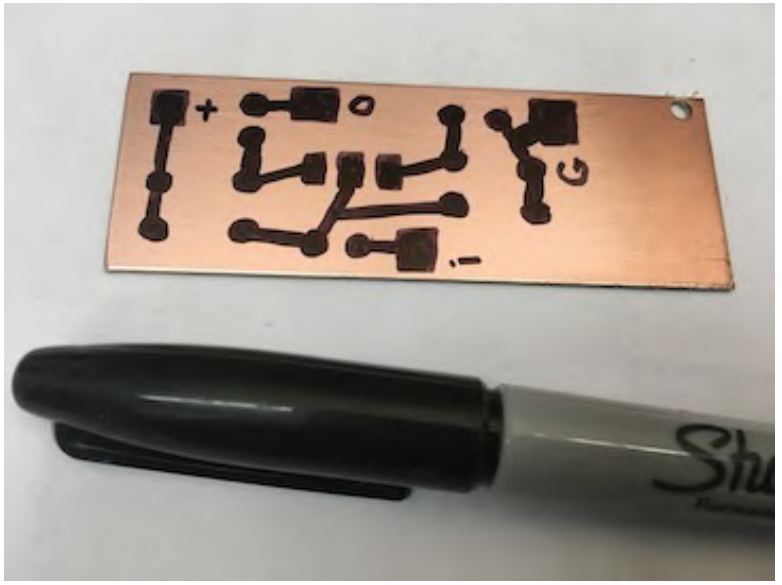
Design the pattern first

- Sketch on grid paper, scaled up, to figure out where the traces should go. Use actual parts to set pad spacing.



Draw on CLEAN AND SHINY copper clad

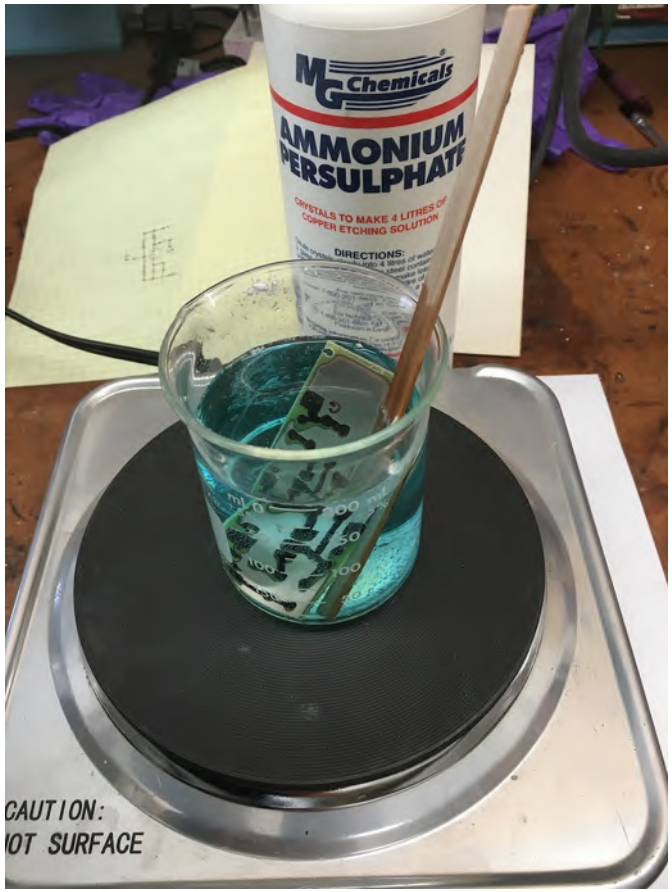
- Ink serves as an **etch resist**. Copper will stay under the ink.
- Get a **new** Sharpie, not that old worn-out, dried-up one
- India ink also works just fine. Use technical pens (e.g., Rapidograph).
- Make sure you have 100% ink coverage with no thin spots
- Corrections and fine details can be made with a hobby knife



Etch your board

- Common etchants are ferric chloride and **ammonium persulphate**. I prefer the latter because it's initially clear.
- Mix the solution per directions in a nonmetallic container. It's usable for many small boards, so plan on keeping it for awhile. 250 g/liter, and use hot water.
- Agitate! Stirring is very important. Suspend the board by a string or stainless steel wire if possible.
- Watch carefully for etch completion. Don't over-etch because it can eat thru your ink resist.

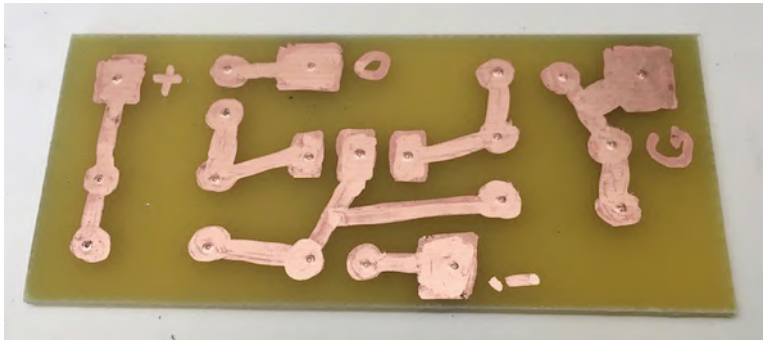
Please dispose of solution as hazardous waste!



About 20 minutes...

Clean up and drill holes

- Remove ink with solvent or steel wool.
- Center punch hole locations
- You will need very small drills. Most holes are #60 or smaller.
- Spin them with a Dremel tool. Drill press fixture is recommended.



Search for **mini drill sets** at Amazon

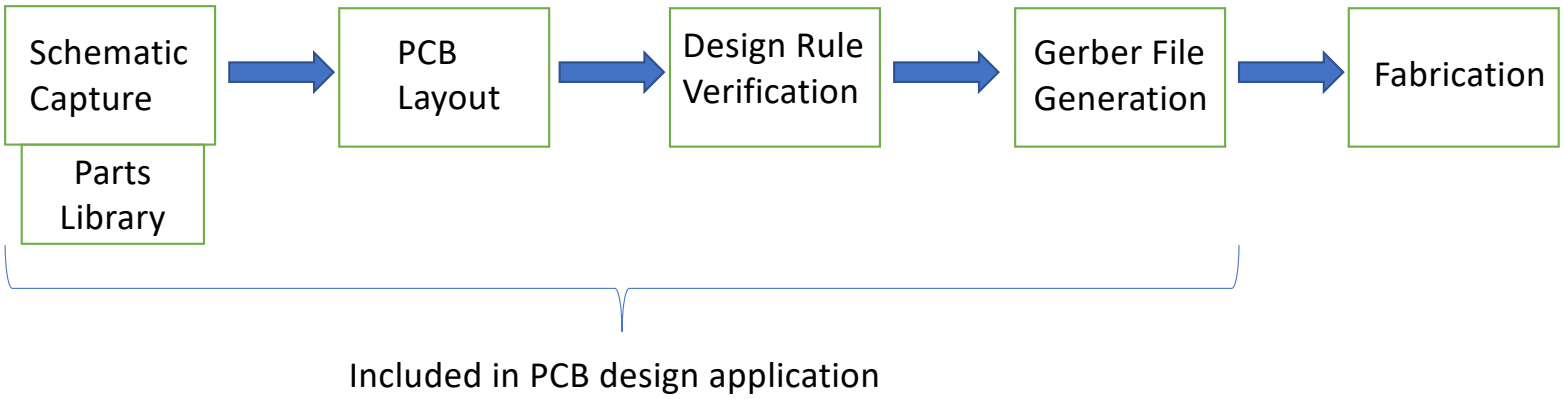
Some other ways to make homebrew PCBs... See YouTube for directions

- Heat transfer
  - Laser print PCB pattern on special **toner transfer paper** amazon link $34
  - Press tightly against clean copper clad, heat in toaster oven (or iron on)
  - Tricky to get 100% transfer especially with fine patterns
  - When it works, it works great
- UV transfer
  - UV-sensitive resist on copper clad—commercial, or spray-on coating
  - Print PCB pattern on transparent film, lay on the board, expose to UV, soak in the appropriate developer solution.
- Turn a 3D printer into a CNC milling machine! (see YouTube)

Note that all of these methods require a PCB layout, using some software

The spiffy way: PCB design software and a commercial fabrication house

Overview of the **workflow**



- Advantage: Commercial-quality product is possible, and in quantity
- Disadvantage: Learning curve on software; time-consuming

Some free/low-cost software options... There are many more, these are popular:

- KiCAD (free)
- Eagle (free but limited to small double-sided, else $60/month)
- DesignSpark (free)
- CircuitMaker (free), now part of Altium 365
- ExpressPCB and PCB123 (free) locked to their own fabs.
- There are countless reviews on the web. I've tried many of these apps and have also used high-end professional software.

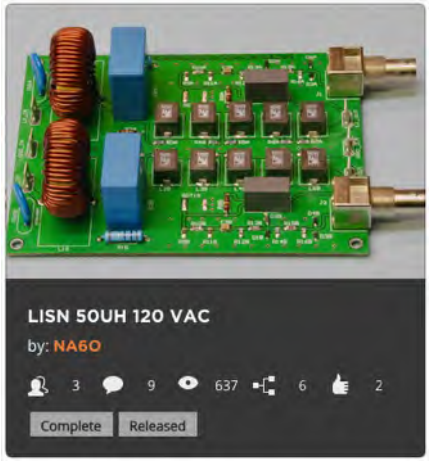
- **CircuitMaker** is my choice. Let's take a quick look...

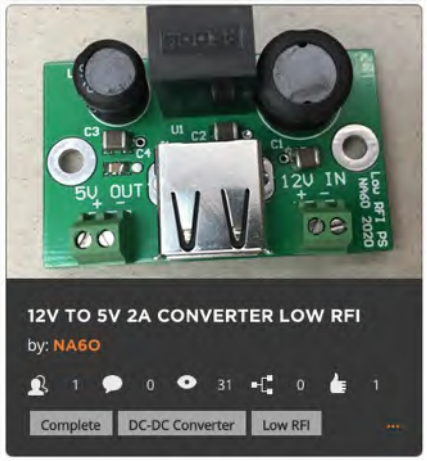
What is *CircuitMaker*?

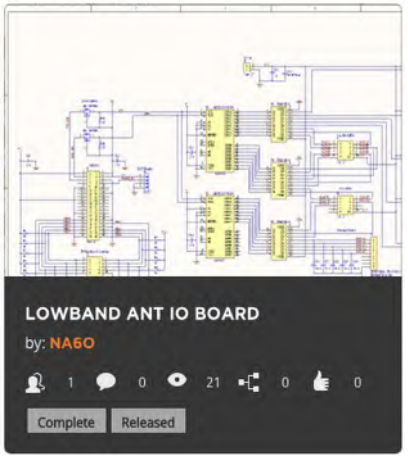
- A free, community version of Altium Designer, a full-featured professional ($4k/yr) electronic design automation (EDA) package
- Includes schematic capture and integrated PCB design
- Application runs under Windows
- Public sharing via *Altium 365*
- Extensive component libraries
- Possibly the best user interface of *any* such application, very modern
- Online documentation, demos, and forum

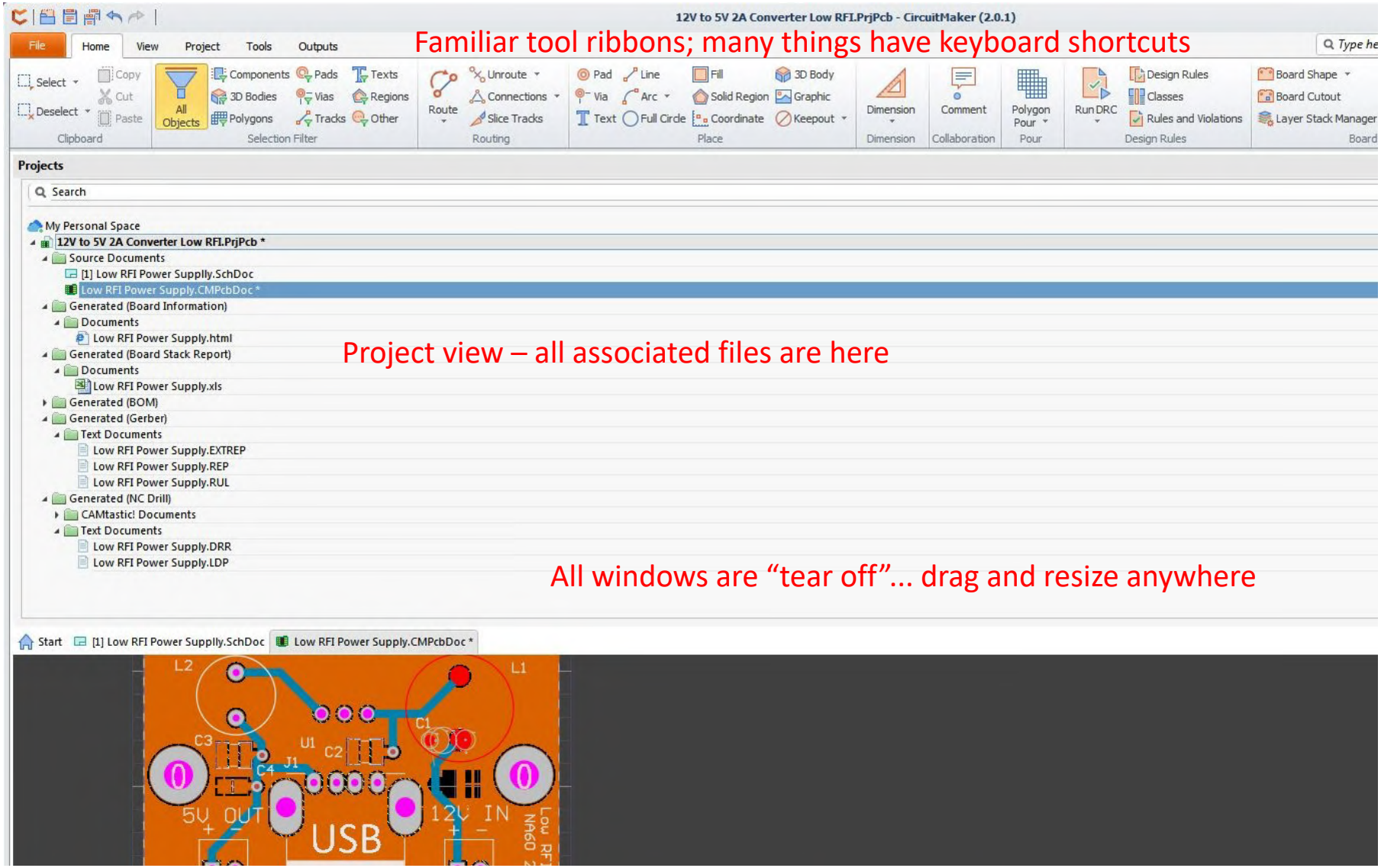
Getting started

- Sign up for free account, you need it to log in every time
- Download and install the app. Watch tutorial videos on YouTube.
- Search existing projects... You might find something close to what you're planning. Fork (clone) an existing design, if you like.

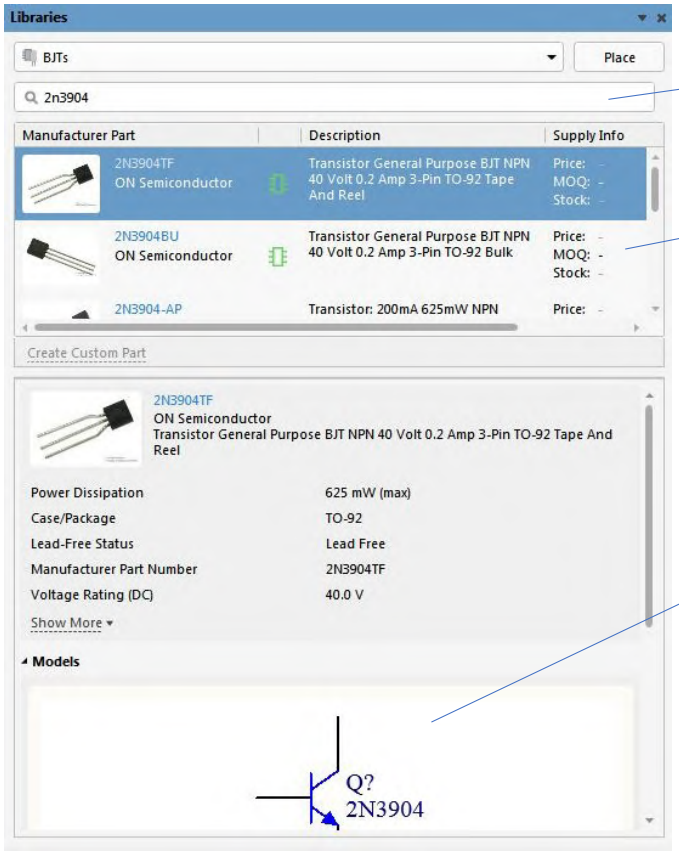







12V to 5V 2A Converter Low RFI.PrjPcb - CircuitMaker (2.0.1)
Familiar tool ribbons; many things have keyboard shortcuts
File
Home
View
Project
Tools
Outputs
Select
Deselect
Copy
Cut
Paste
Clipboard
All Objects
Components
3D Bodies
Polygons
Pads
Vias
Tracks
Texts
Regions
Other
Selection Filter
Route
Unroute
Connections
Slice Tracks
Routing
Pad
Via
Text
Line
Arc
Full Circle
Fill
Solid Region
Coordinate
3D Body
Graphic
Keepout
Place
Dimension
Comment
Polygon Pour
Run DRC
Design Rules
Classes
Rules and Violations
Board Shape
Board Cutout
Layer Stack Manager
Projects
Search
My Personal Space
12V to 5V 2A Converter Low RFI.PrjPcb *
Source Documents
[1] Low RFI Power Supplly.SchDoc
Low RFI Power Supply.CMPcbDoc *
Generated (Board Information)
Documents
Low RFI Power Supply.html
Generated (Board Stack Report)
Documents
Low RFI Power Supply.xls
Generated (BOM)
Generated (Gerber)
Text Documents
Low RFI Power Supply.EXTREP
Low RFI Power Supply.REP
Low RFI Power Supply.RUL
Generated (NC Drill)
CAMtastic! Documents
Text Documents
Low RFI Power Supply.DRR
Low RFI Power Supply.LDP
Project view – all associated files are here
All windows are “tear off”... drag and resize anywhere
Start
[1] Low RFI Power Supplly.SchDoc
Low RFI Power Supply.CMPcbDoc *
L2
L1
C1
C3
U1
C2
J1
C4
5V OUT
USB
12V IN

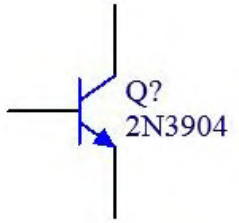
Parts libraries are truly vast. And you can create, dupe, or edit anything.



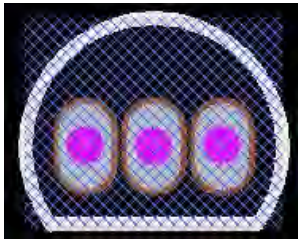
- Search by part number, stock number, etc.
- Linked to manufacturer's and supplier's websites
- Not everything has a model. Parts can't be placed without creating one first.

Every part must have a **model**

1. A symbol for the schematic

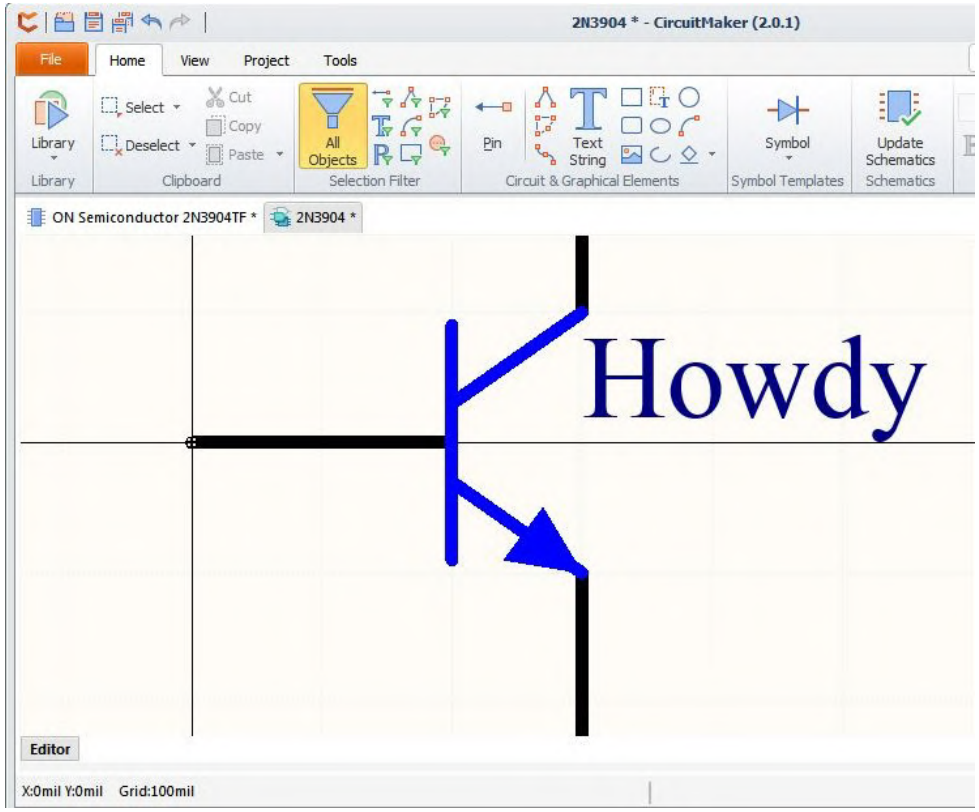


2. A footprint for the board layout
   Optionally, a 3D version as well



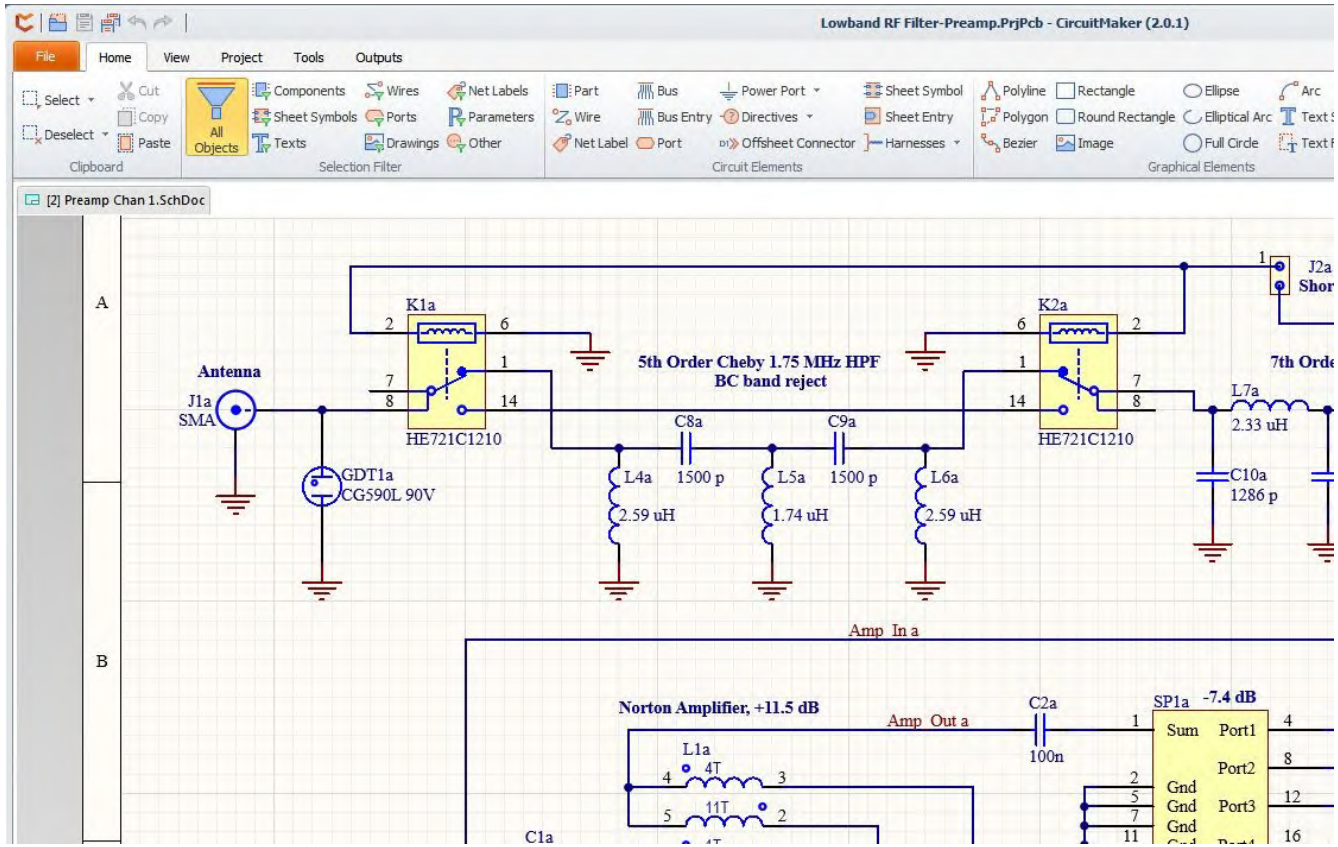
3. If you had the “real” version of Altium, you could also have a SPICE model for simulation

The component editor creates models



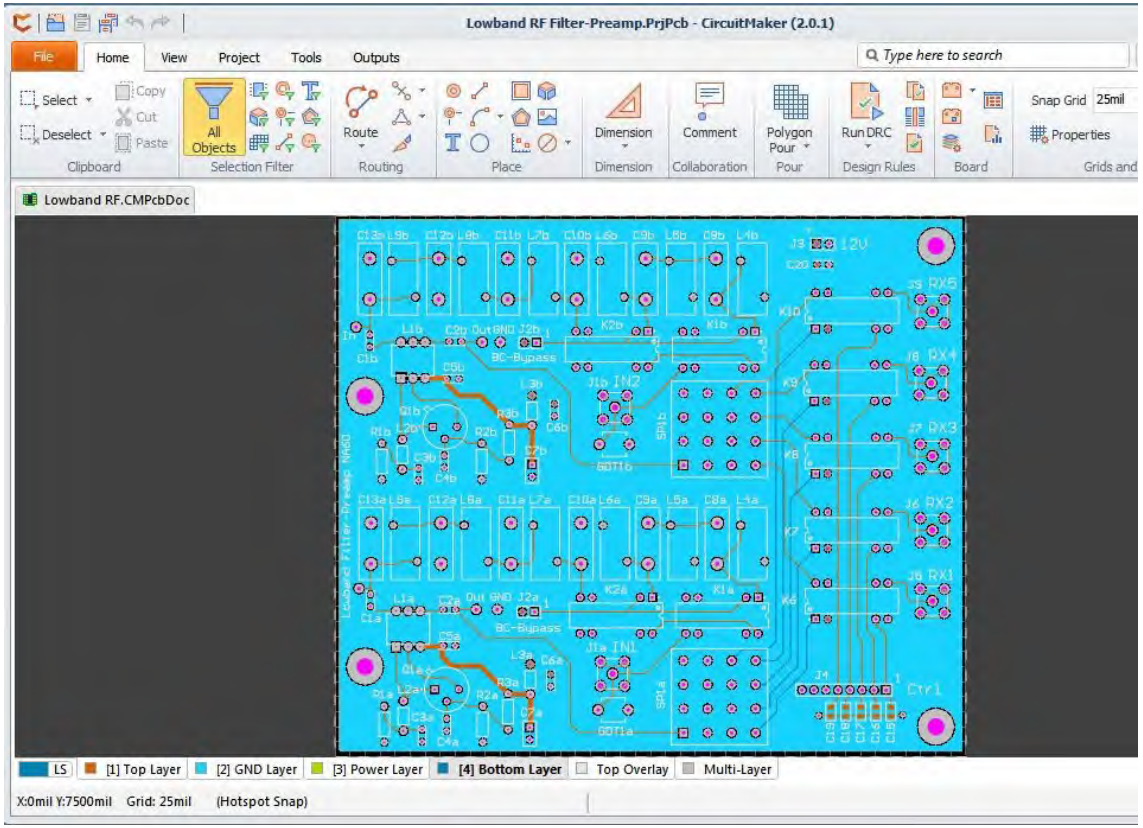
- Use simple tools to build symbols and footprints
- Start with standard ones, or copy from other existing parts
- Save in your own library, and/or make it public

Draw the schematic first. I work from a manual sketch, building parts as I go



- After building the schematic, you *update the PCB*.
- This transfers data via a *netlist* file to the PCB editor.

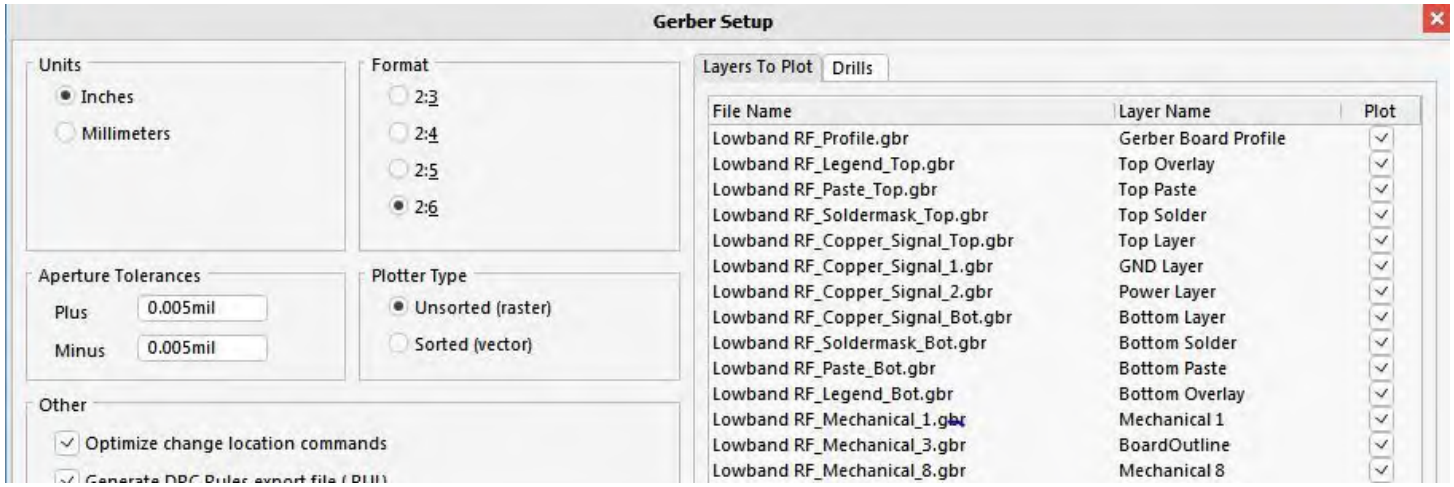
Your inner artist comes out when designing a PCB. Autoroute is… semi-useful.



- Parts *placement* is crucial
- Manual routing is usually best
- Many settings to learn about, like trace widths
- Keep your wits about you regarding what layer you're on!

Once the PCB layout is done, you can check and export your design

- **Design Rules Checking** looks for all kinds of flaws, like object too close for fabrication, missing connections, etc. Highly configurable... By default it catches a lot of really minor things.
- Finally, you generate the **Gerber plots** for fabrication. Most fab houses have instructions on what files and settings you will need.



Gerber files go to a fabrication house

- Zip up your Gerber files and get ready to upload...
- Many fab houses to choose from. The Chinese ones are incredibly cheap and fast.
- I use JLCPCB. Good website, instant quote, reliable, fast DHL shipping.
- Example: 2-sided, 4 in square, 5 pcs: $2 plus $20 shipping. I usually get my order in 4-5 days! Quality is excellent, never an issue.
- Made-in-USA OSHPark is popular. Similar order will cost about $90 and takes 9-12 days.

Typical fabrication choices

- Board color 

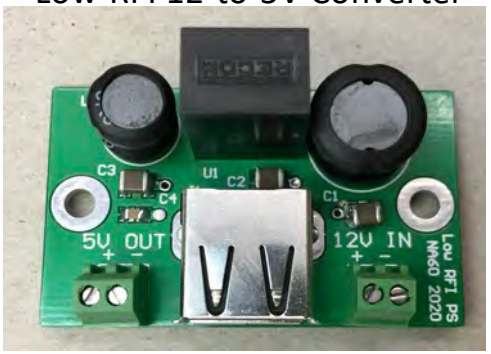
- Plating 

  - HASL = heat activated solder leveling; lead or lead-free
  - ENIG = electroplated nickel-gold (more expensive)
- Copper weight 

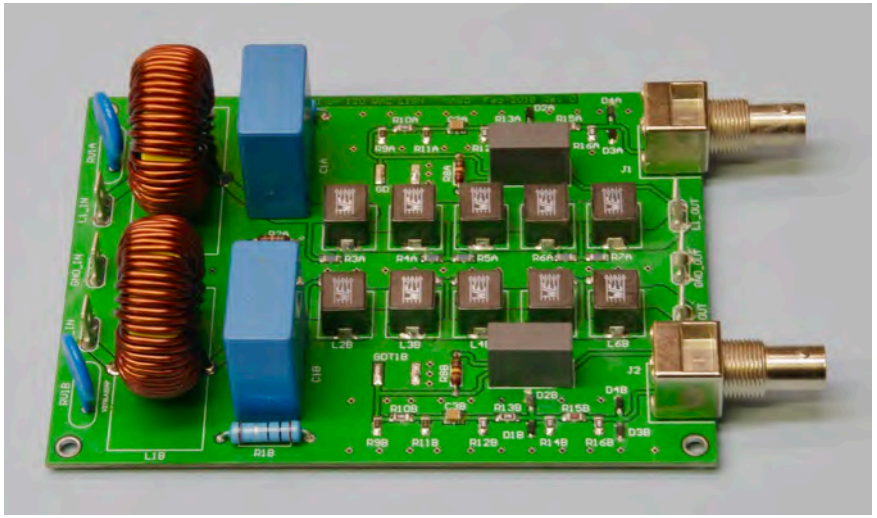
  - Normally 1 oz.

Some finished projects, using CircuitMaker and JLCPCB

Low-RFI 12-to-5V Converter



Line impedance stabilization network (LISN)



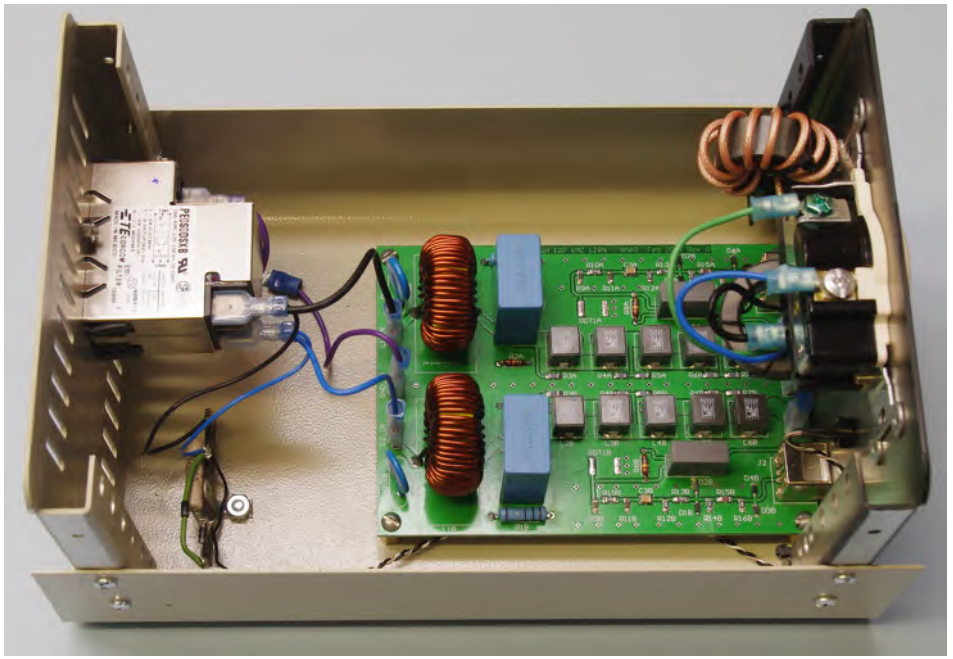
Lowband RX antenna filter and distribution

Most projects need an enclosure. Some mechanical fabrication may be required...

Hammond cast-aluminum enclosures (1590 series), many sizes, also gasketed, flanged. Or plastic. All are very easy to drill.

$15-$30 typical at Mouser, Digikey, etc.

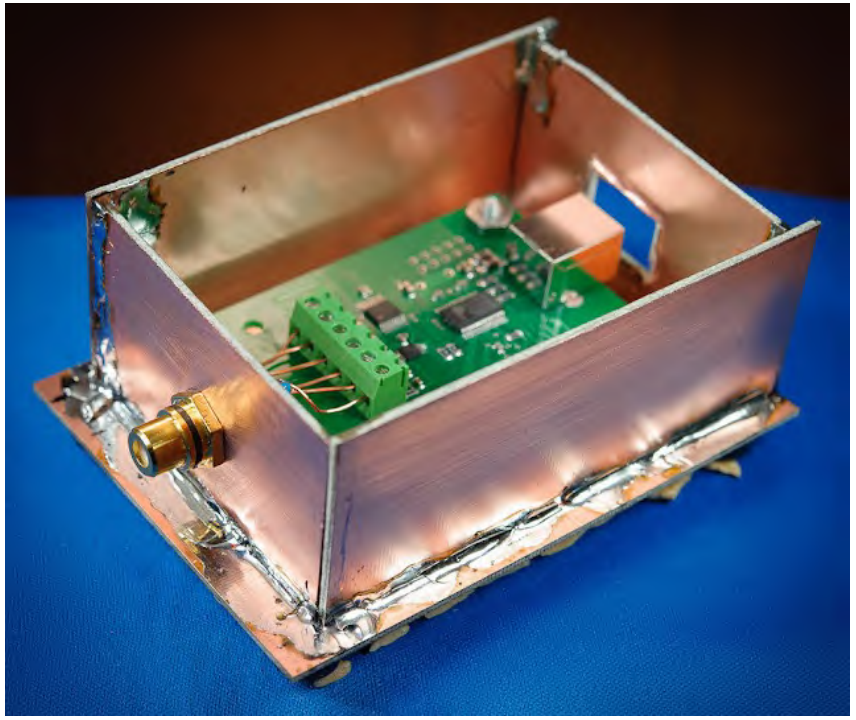
Hammond or Bud instrument enclosures



Safety note: If you build anything with over 48 V involved, please have your work reviewed by someone qualified.

You can make a cheap, custom enclosure from copperclad PC board material

- Cut with tinsnips, or better with a tablesaw if you have one
- Drill or cut openings
- Solder joints. Sometimes a larger soldering iron or soldering gun works faster.
- Coat with clear acrylic spray when you're done if you want to preserve the shiny copper look



What tools do I need for simple enclosures?

Combination square

Files, round and flat

Coping saw for square holes

Caliper

Automatic center punch

Scriber

Step Bits, especially for large holes and thin sheet metal

Nibbling tool for square holes

... and a cordless drill and regular drill bits

Labels for panels

- Free-hand pen (yuck), dry-transfer letters (laborious), Brother label maker (quick & tidy), or print on transparent adhesive film (nice!)

"Clear sticker film" or "Repro film" at Amazon

If it’s going outside, get a weatherproof enclosure with gaskets!

References

- ARRL Handbook
- *The Art of Electronics* by Horowitz and Hill
- And if you can find it and afford it,
  - *Experimental Methods in RF Design* by Hayward, Campbell and Larkin