Printed Circuit Boards

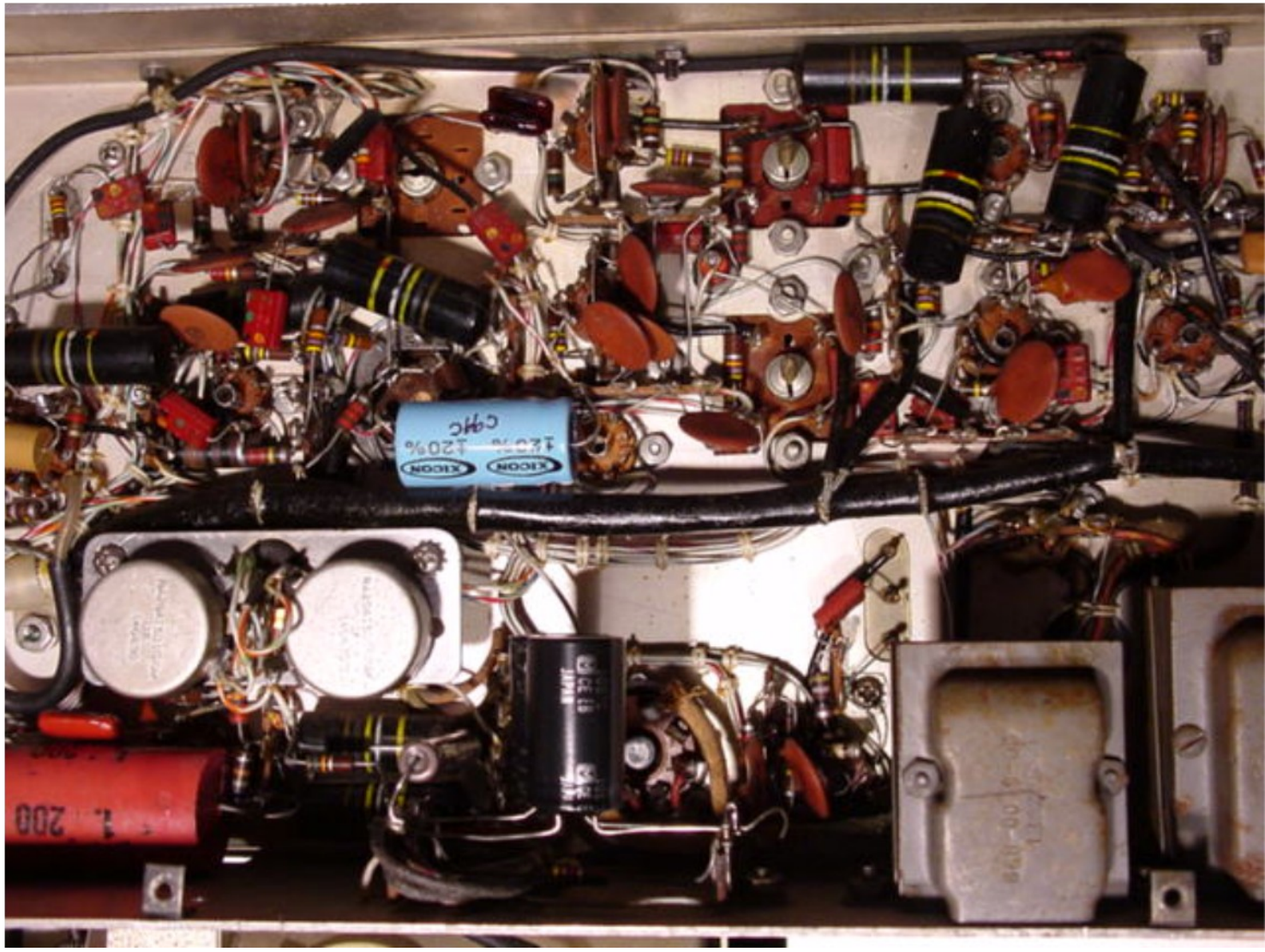
Layout with Kicad, and Ordering

Dave - AI6VX

Before Printed Circuit Boards (PCBs)

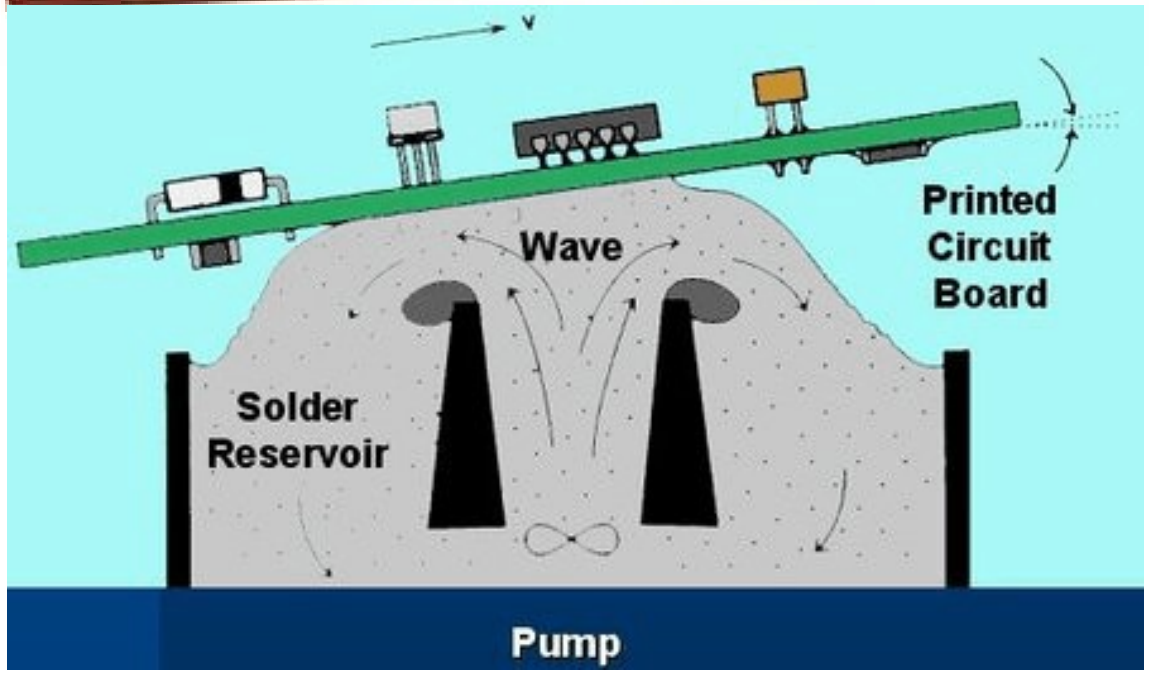
Point to Point Wiring

- Components wired to each other, tube socket lugs or terminal strips
- Required a lot of touch labor
- Assembler skill affected product quality
- Hard to mass produce



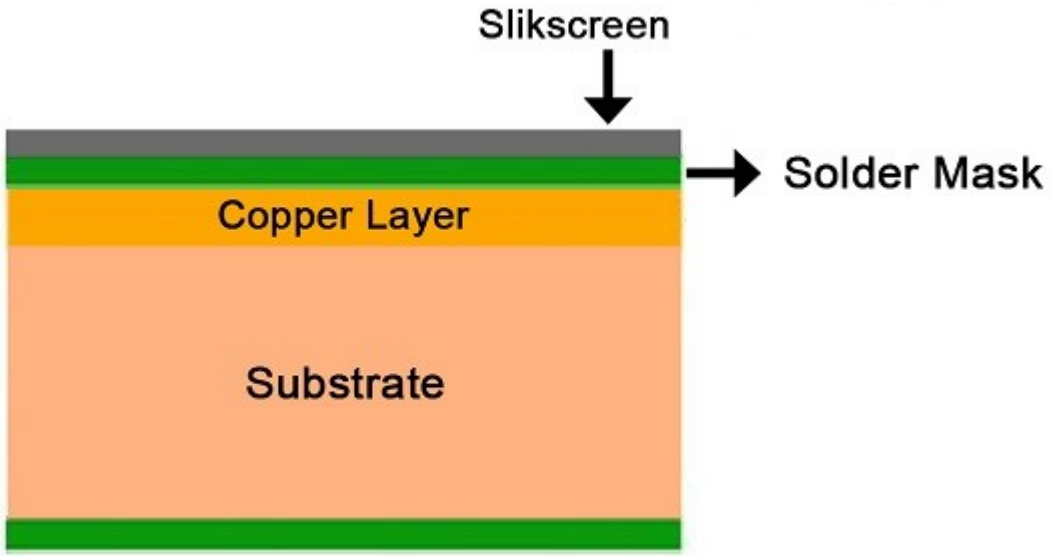
Why PCBs?

- Consistency
  - Part placement, lead length.
- Density
- Manufacturability / Automation
  - Wave soldering would solder an entire board in seconds
- Less operator skill required

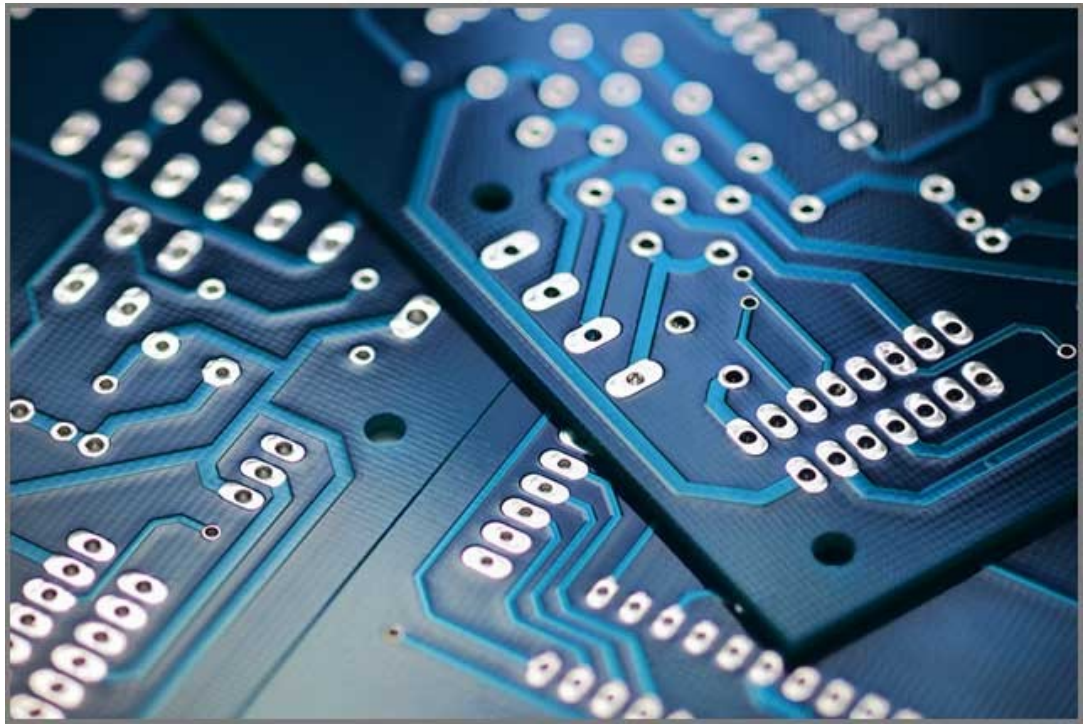


What is a PCB ?

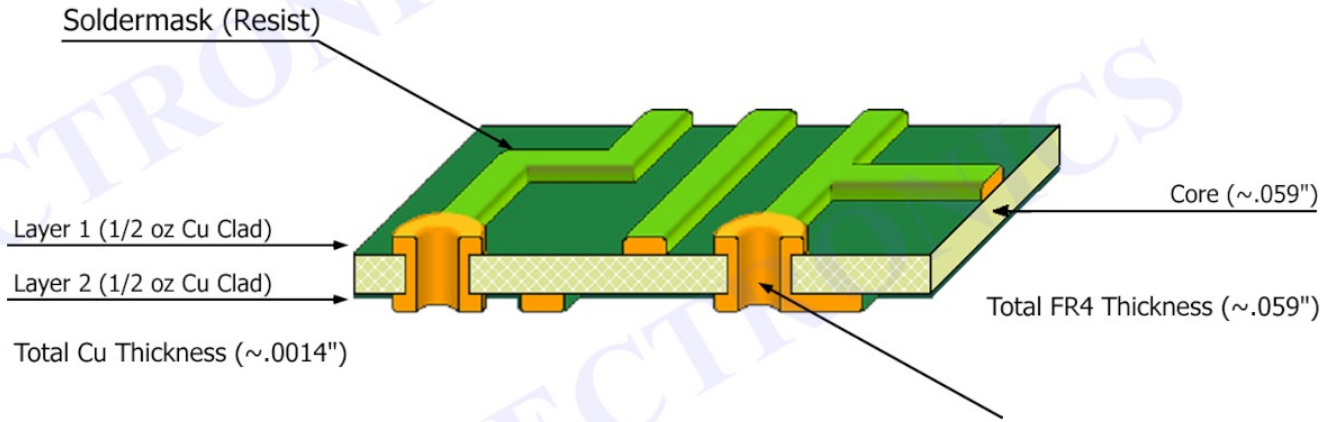
Method to wire components together



- Substrate – Insulated mechanical support
  - Phenolic, FR4 (glass epoxy), etc
- Conductive Traces – Usually copper
  - Can be on both sides of the substrate, or even inner layers (multi-layer boards)
- Solder Mask – Thin insulator covering board except for pads
  - Prevents shorting between pads & traces
- Silkscreen – Legend marking



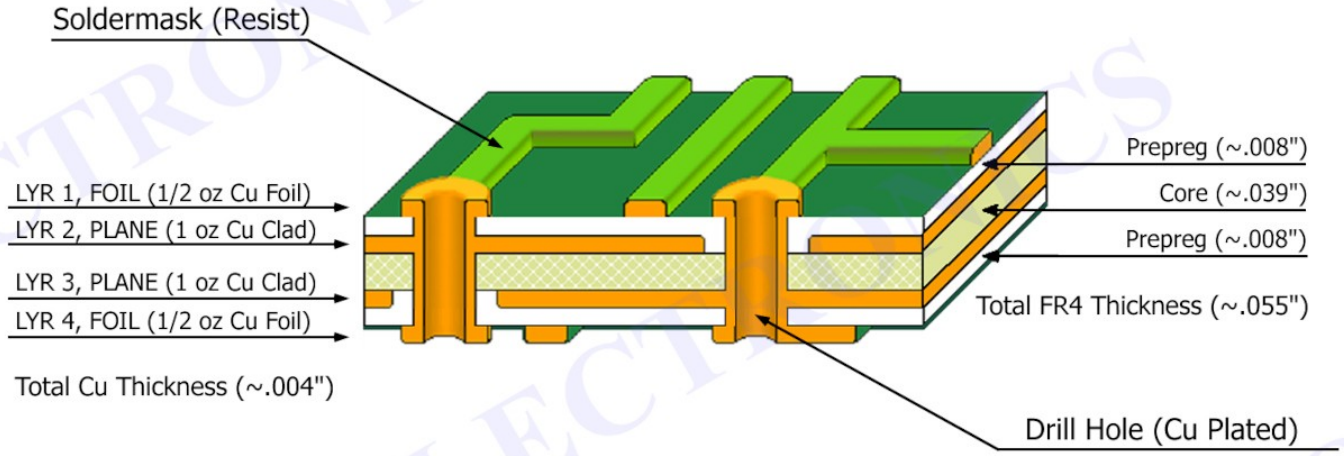
Double Sided and Multi-Layer Example



Two Layer

Stack Height (thickness) ~0.061"
+ Cu plating (post drill) ~0.002"

= Overall Thickness ~0.063"



Four Layer

Stack Height (thickness) ~0.059"
+ Cu plating (post drill) ~0.002"

Can I Make Printed Circuit Boards at Home?

Yes! Single sided can be quick and easy

- Hand drawn, toner transfer, photographic artwork
- Double sided is hard
  - Aligning top and bottom artwork
  - Connect ‘vias’ manually (top trace to bottom trace)
- Etching is dirty
- Drilling boards can be difficult
- Hard to create small traces and small spacing
- Usually have raw exposed copper. No solder mask, no silkscreen

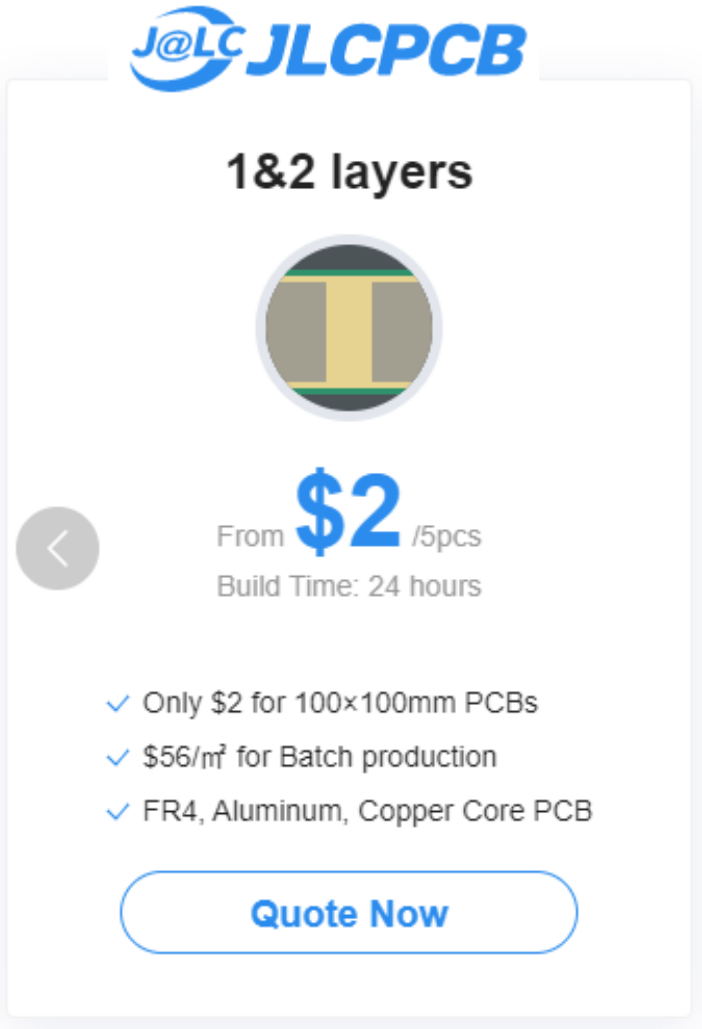
Professionally Made is Expensive Right?

No!

Offshore vendors provide high quality ‘full service’ boards at very good prices.

- Quick fabrication time
- Shipping speed greatly affects total cost and delivery time.

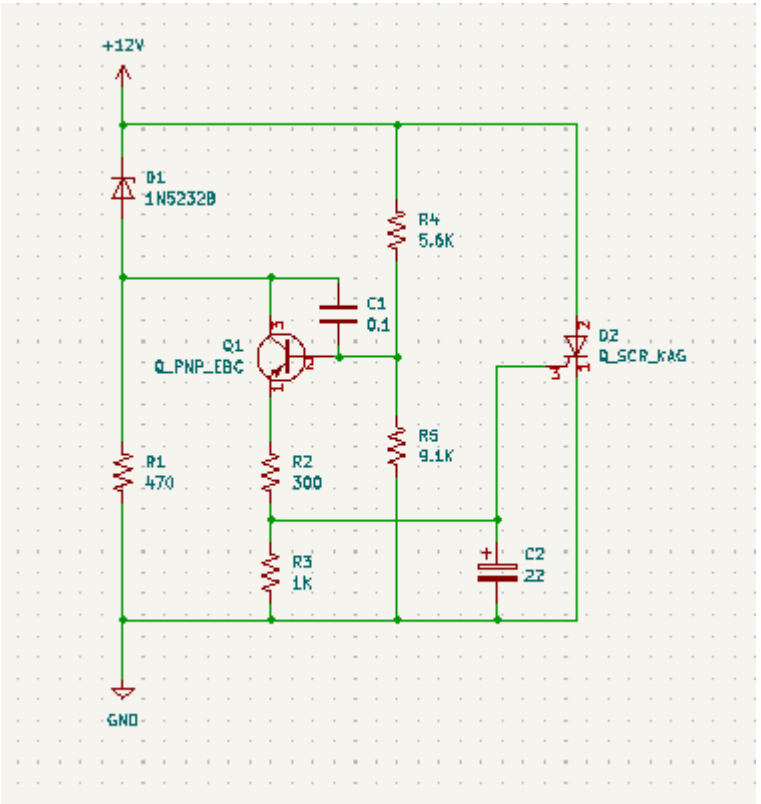
For less than $10, you can have professionally made PCBs (yes, plural) delivered to your door in 2-4 weeks.



PCB Creation Steps – Schematic Entry

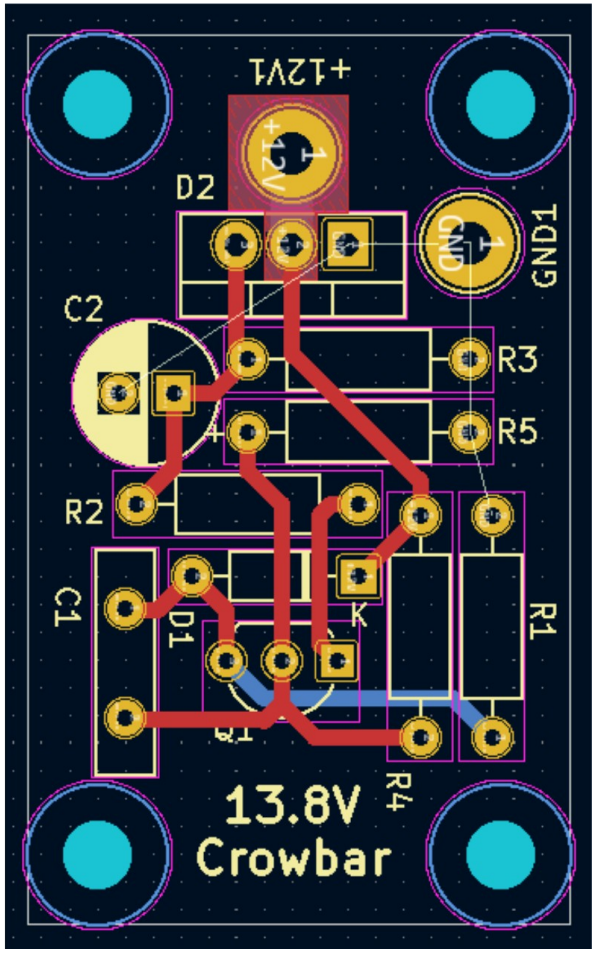
Best to start with a schematic

- Creates the component list with IDs
- Assigns a footprint to the component (size of components and pad positions)
- Generates a Net List (list of how components and signals are connected together



PCB Creation Steps – Board Layout

- Arrange component footprints on workspace
  - ‘Rubber bands’ show connections
- Connect matching terminals and signals together
  - Size traces for amperage
  - Keep minimum clearances to other traces and pads
- Set mounting holes
- Set board size
- Perform design rule checking
- Export Gerber files



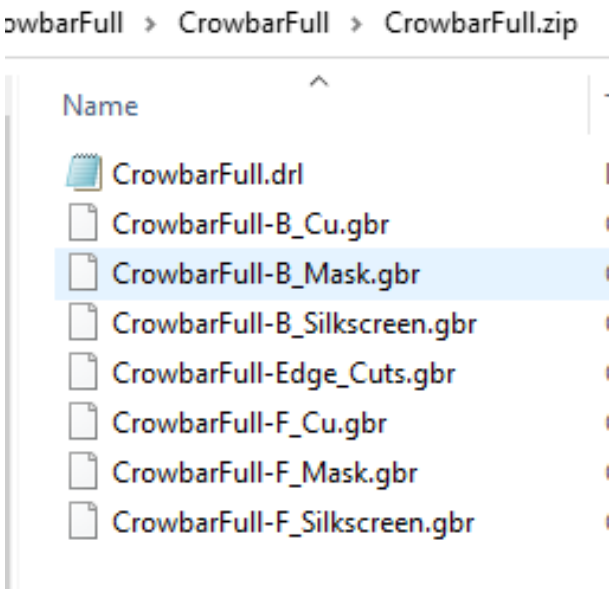
Important !

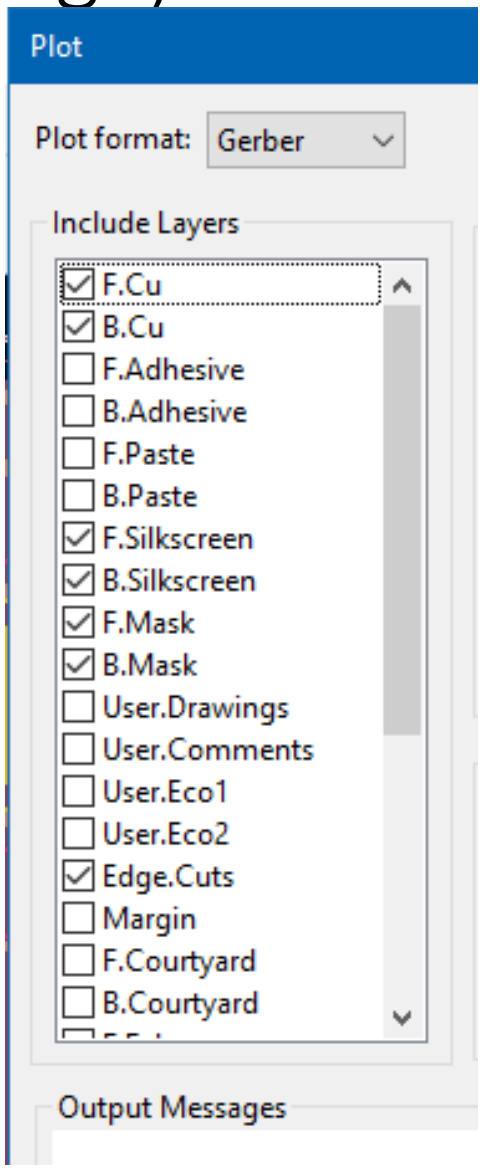
- All Layers are viewed from the top

PCB Creation Steps – Build Files (Gerber Package)

Files needed for the board house to produce the circuit board

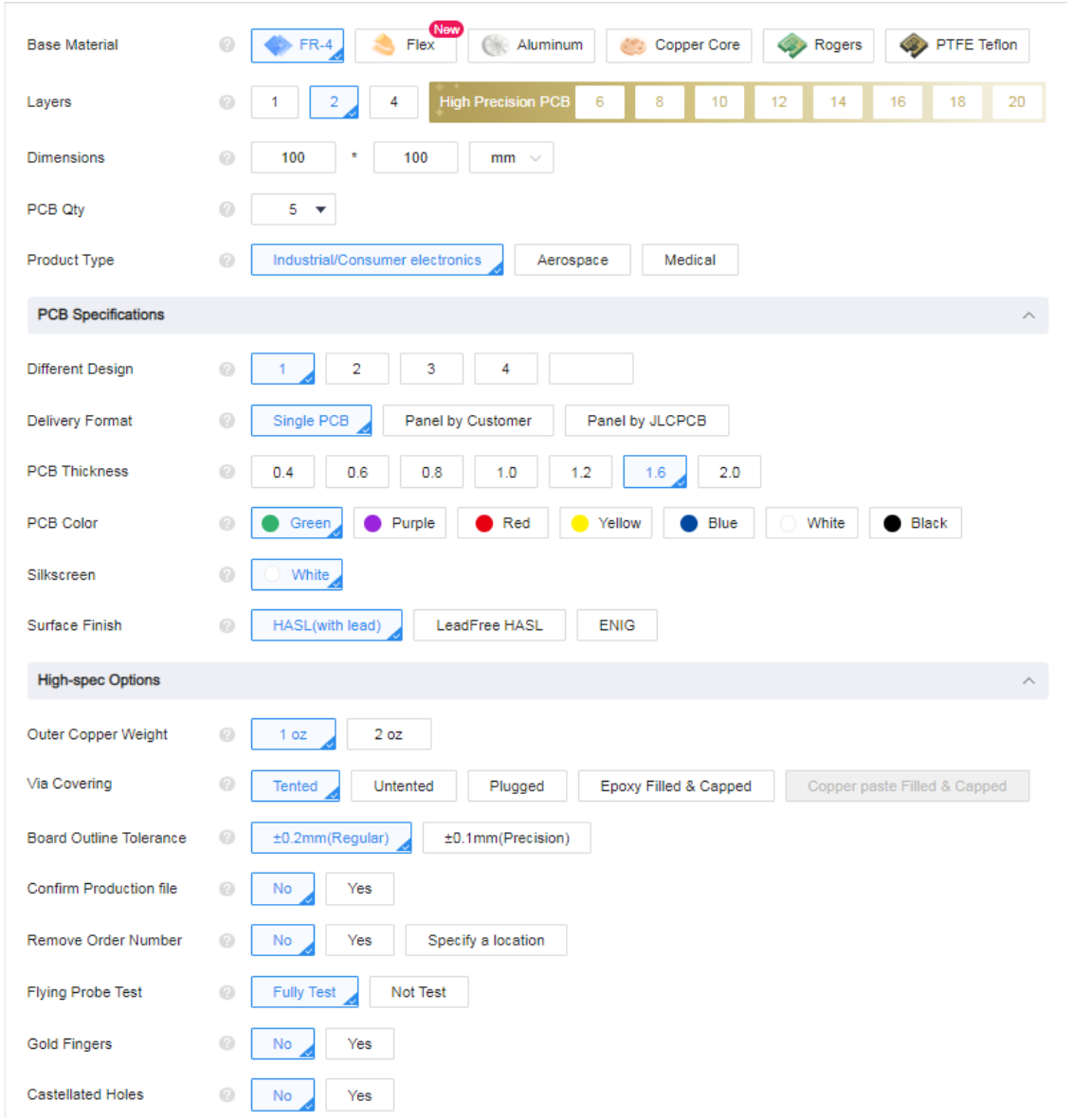
- Copper Layers (top, bottom, middle1, middle 2…)
- Solder Mask (top, bottom)
- Silk Screen (top, bottom)
- Solder Paste (top, bottom)
- Edge Cuts (board outline)
- Drill File (plated, un-plated)



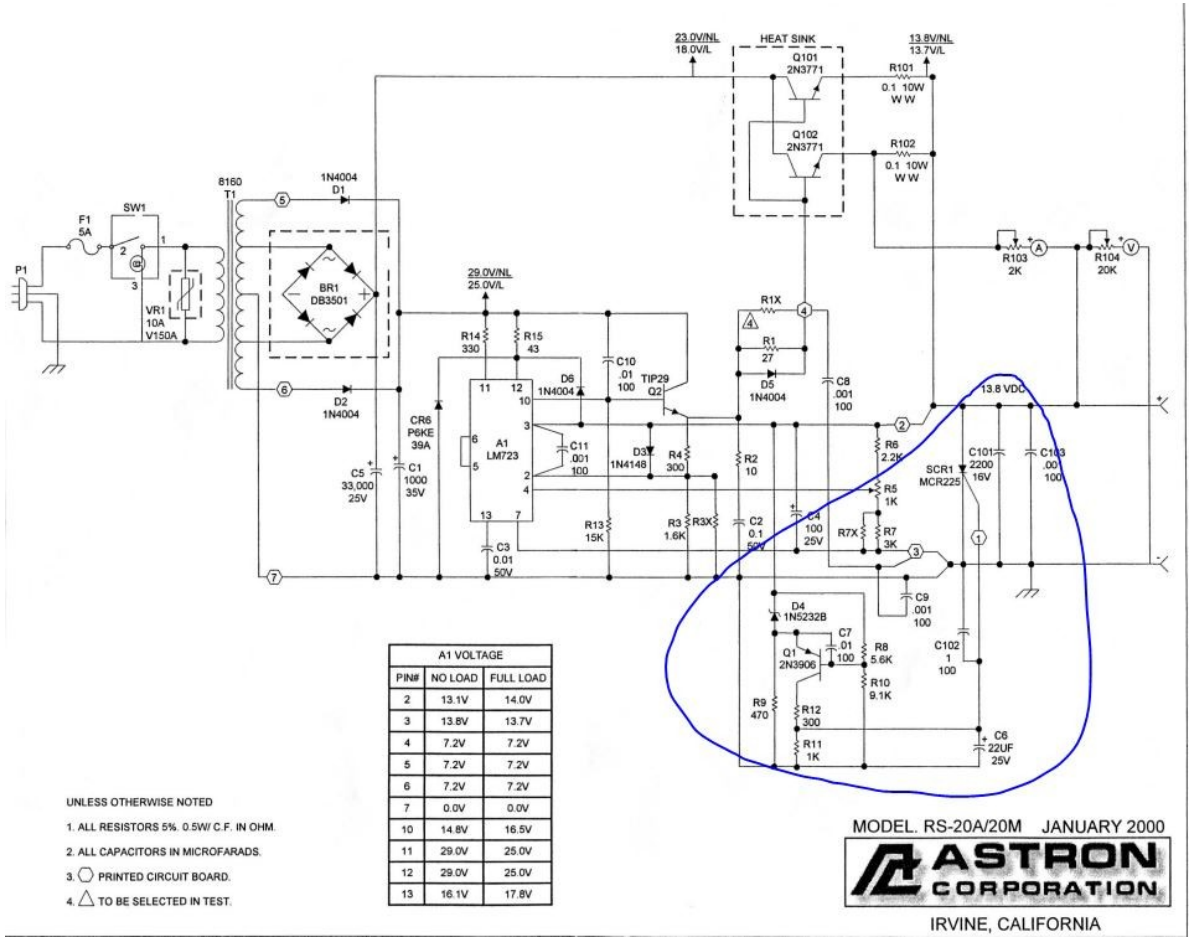


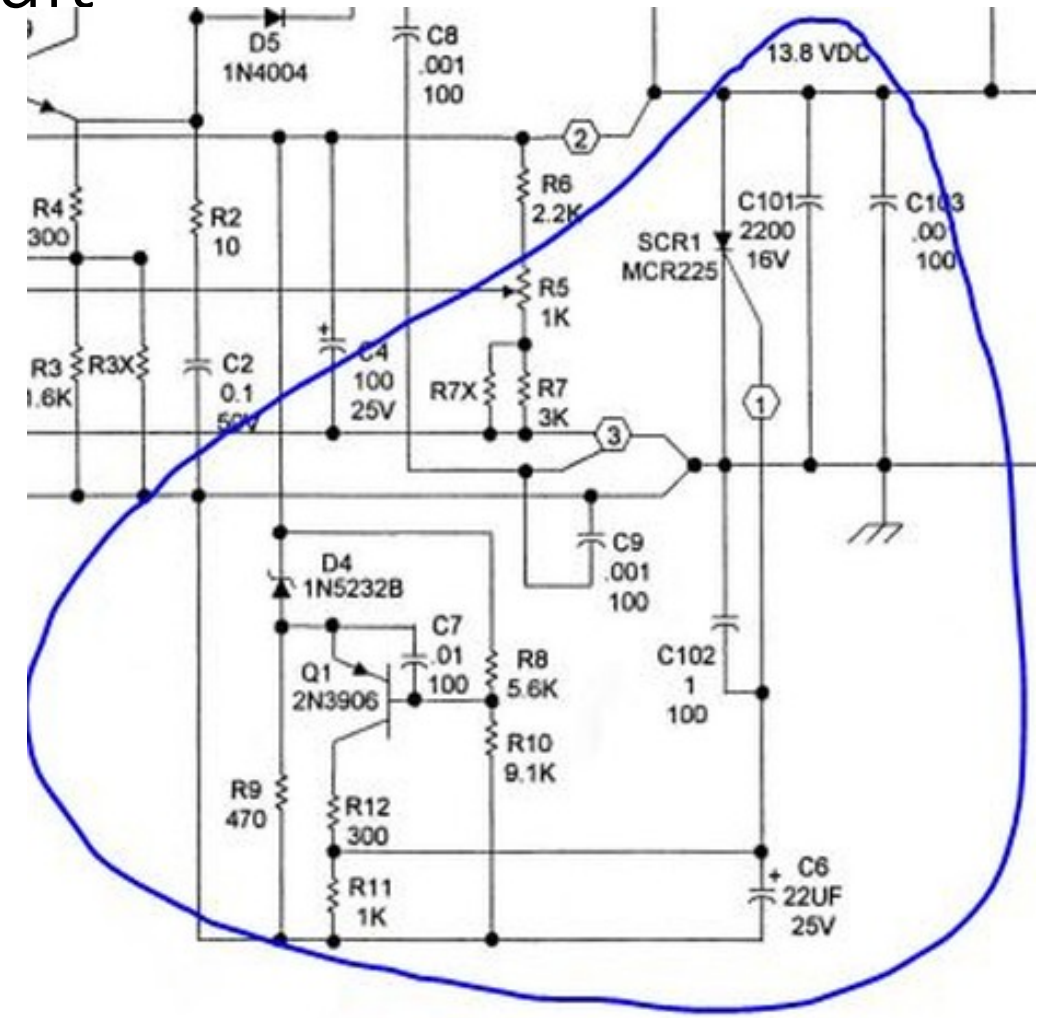
PCB Creation Steps Ordering Boards

- Size
- Number of Layers
- Substrate type and thickness (FR4 1.6mm most common)
- Copper Weight (usually 1 oz)
- Solder Mask Color (usually green)
- Finish
  - HASL – hot air solder leveled (not lead free)
  - ENIG – electroless nickel gold (lead free, additional cost)
- Qty
- Shipping Choice
- Gerber File Set - upload



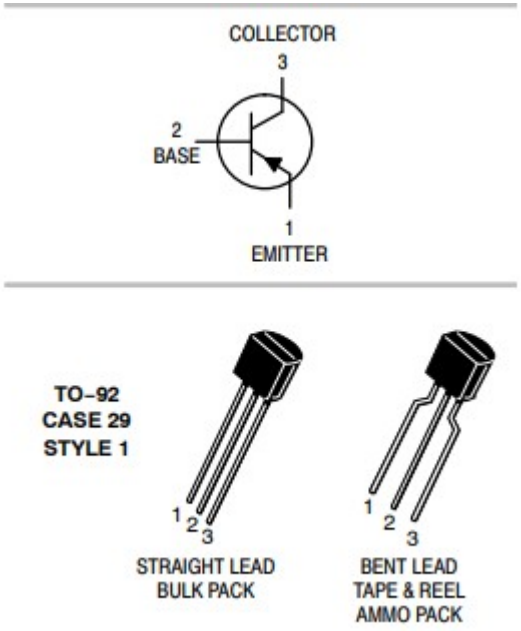
KiCad Example – Crowbar Circuit
(KEE-kad)



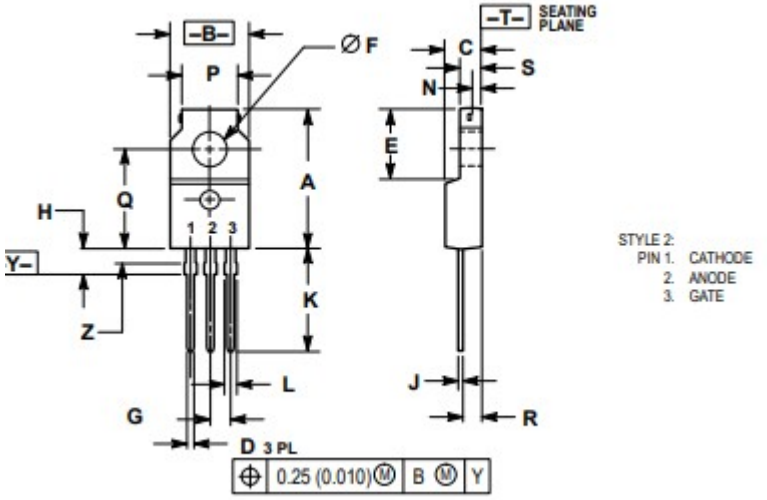


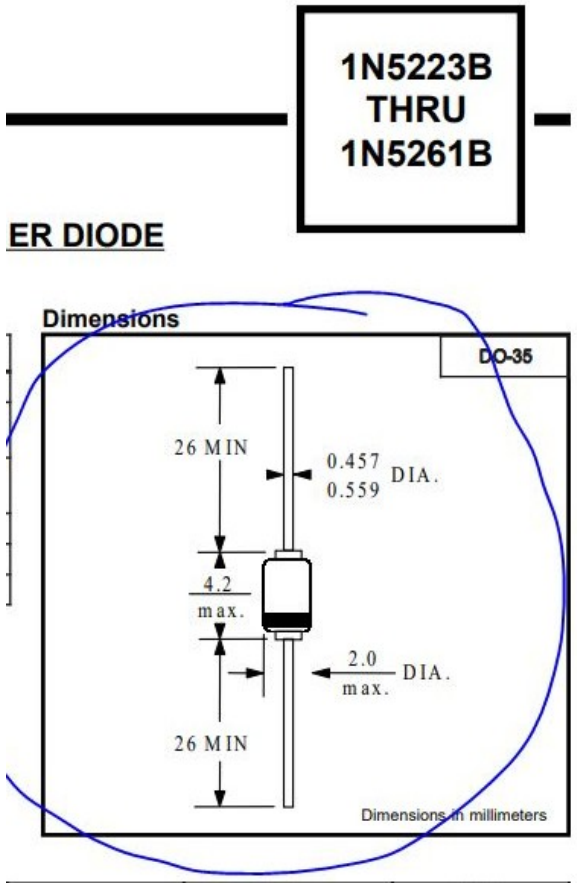
Package size and Pinout

2N3906 TO-92 Case



TO-220 SCR





1N5232B 1W Axial

Live Demo of KiCad, Schematic capture, board layout, gerber and drill file export, and ordering PCB from JLCPCB.

Glossary

- Via – a connection from one layer to another, usually a plated drilled hole.
- FR4 – a type of PCB substrate. Also known as ‘glass epoxy’
- Gerber –a text based complete ‘image’ description of 1 layer of a PCB
- Silkscreen – outermost layer on a PCB that shows part location and ID
- Solder Mask – a thin coating that covers the Color (green and black are common)
- HASL – Hot Air Solder Leveling. Coats pads with a thin layer of solder
- ENIG – Electroless Ni-Gold plating. Lead-free option. Pads are finished with gold.
- Minimum line/space – smallest trace width and distance between traces/conductors. Set by PC board manufacturer’s capabilities
- Footprint – A pad location and perimeter model for a component. Includes numbering of pads, which need to match numbering and function for the component in the schematic
- Edge Cut Layer – denotes outer edge of PC board. Used to ‘cut’ board to size.
- Flying Probe Test – electrical testing of board by manufacturer. Used to test continuity and shorts between points.