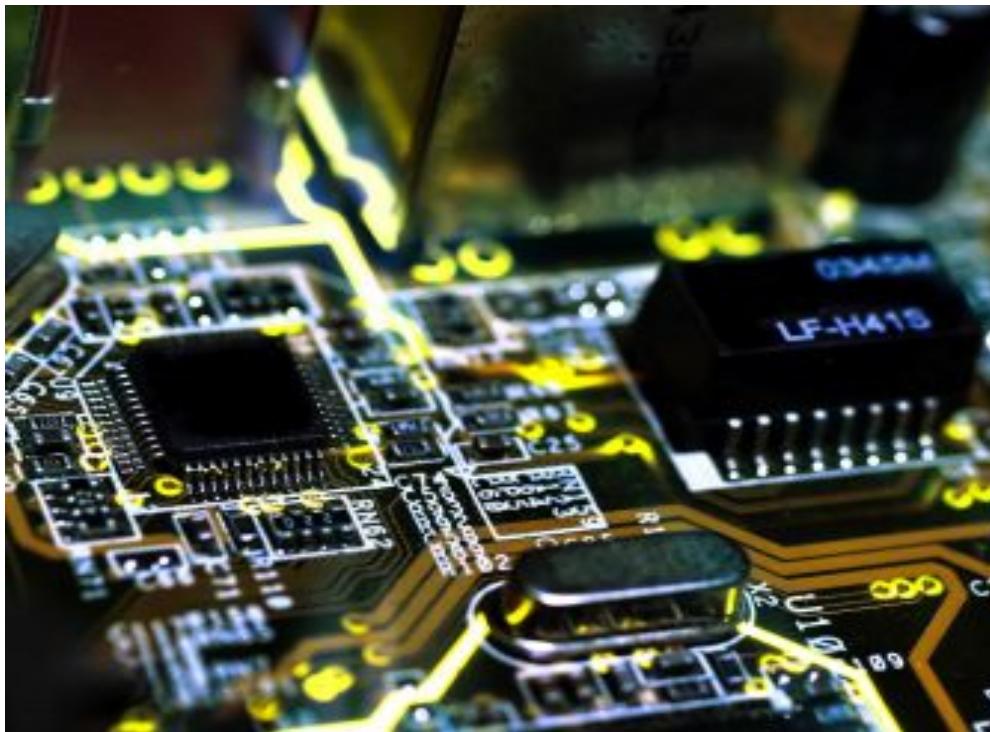


ART of Printed Circuit Board Design

Design-Capture-Prototype
By Manoj Gulati

Printed Circuit Board



3Q's in your Mind

- What is a PCB?
- Why use a PCB?
- How do I make a PCB?

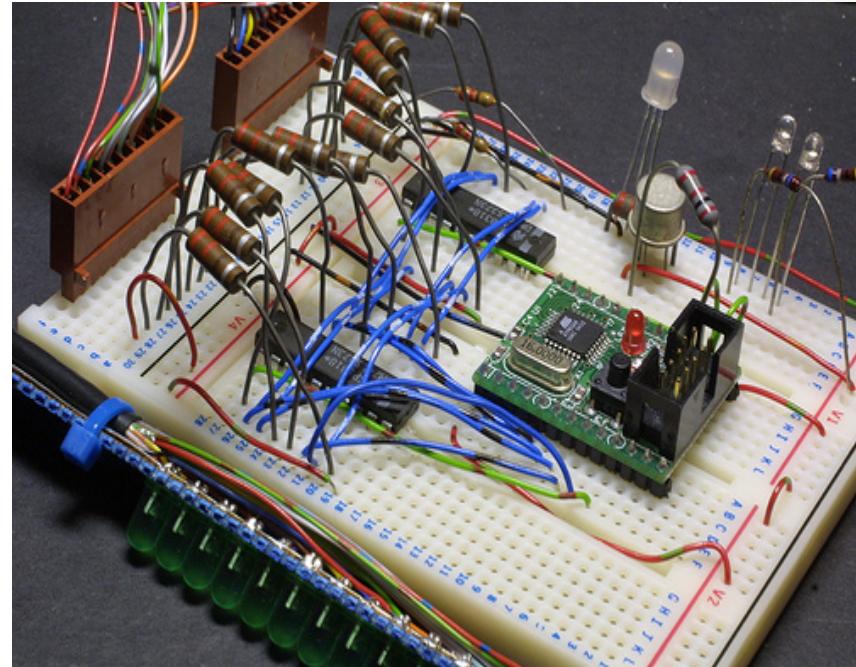
What is PCB?

- Wikipedia: A **printed circuit board**, or **PCB**, is used to mechanically support and electrically connect electronic components using conductive pathways, tracks or signal traces etched from copper sheets laminated onto a non-conductive substrate.
- Simple Definition : An electronic circuit board having copper wire interconnects on non conductive board.

Why use PCB ?

Can't use breadboards?

- Features
 - Surprisingly Affordable
 - Highly reliable
 - Rugged
 - Compact
- Drawbacks
 - Requires more layout than other board types
 - Higher initial cost than wire wrap or point-to-point construction



18/09/15

(5)

Image Courtesy:

<http://dev.emcelettronica.com/breadboard-circuits-tips-tricks>

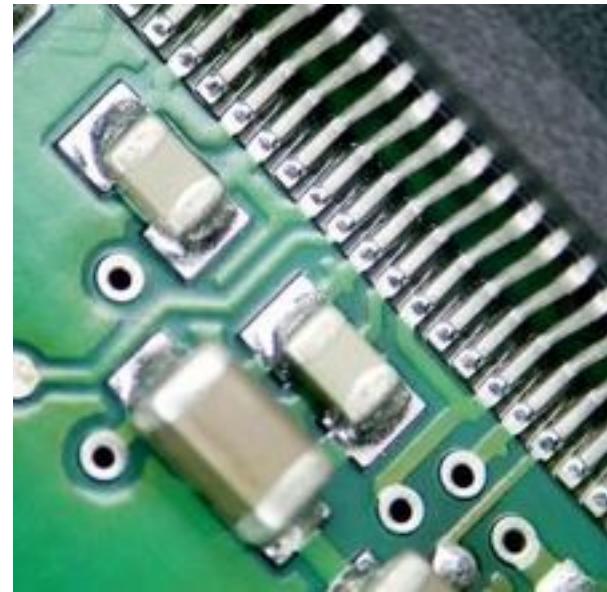
How Do I make PCB?

PCB Design Steps

- 1** Design the circuit
- 2** Place the components
- 3** Route the wires
- 4** Manufacture

Types of PCB's

- Surface mount technology(SMT)
 - Top and bottom planes
 - Minimal routing area
- Through-hole mounted (PTH)
 - All components on same side
 - Reduces routing area on all planes



18/09/15

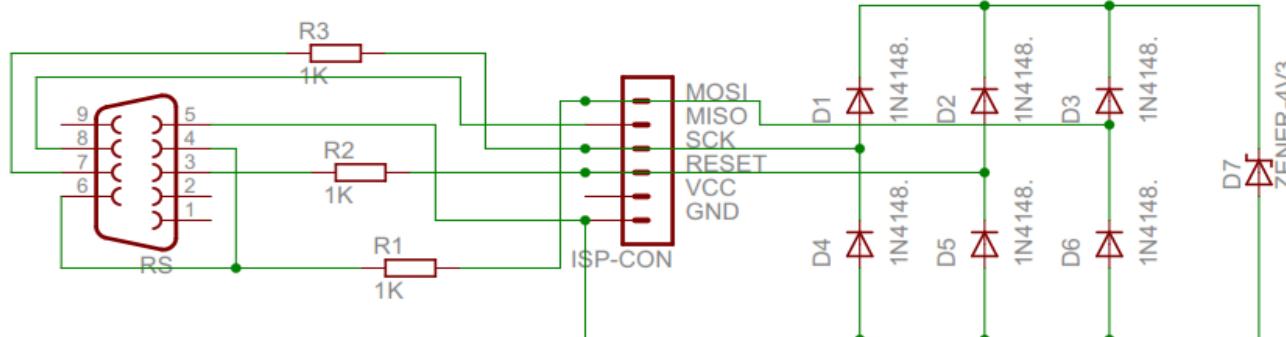


EDA Software

- PCB design software package
 - Design the circuit (Schematics editor)
 - Place the components (PCB editor)
 - Route the wires (Auto router module/ Manual routing)
- Examples:
 - Orcad PCB Capture (Cadence)
 - Mentor Graphics PADS
 - EAGLE CAD (CADSOFT)
 - Tiny CAD
 - Protel
 - ExpressPCB

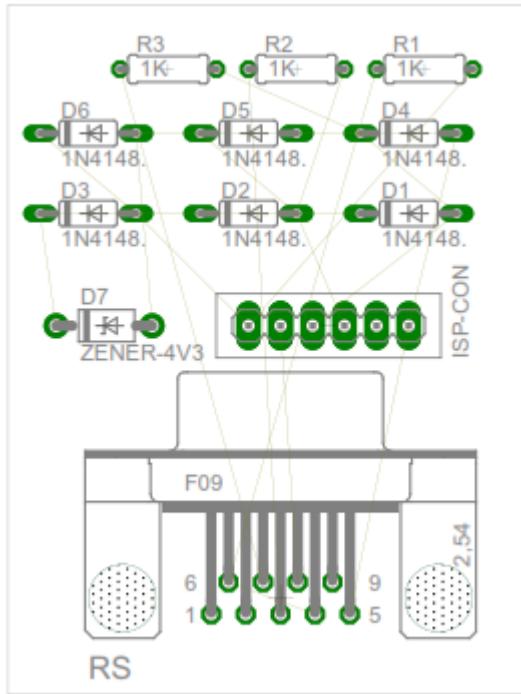
Schematic Capture

- Component Selection
- Package Selection
- Library Part Search
- Pin in/outs
- Interconnections
- Easily Readable
- High-Level Block Diagram

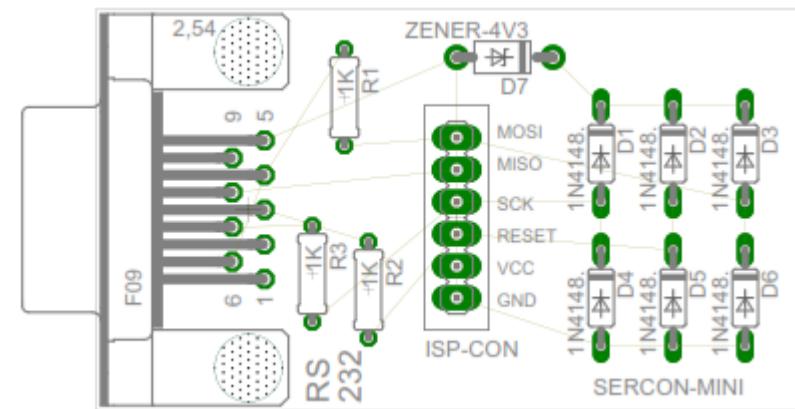


Placement of components

- Place big components first- e.g. microcontroller
- Place components in separate functional groups
- Put a ground plane under all components

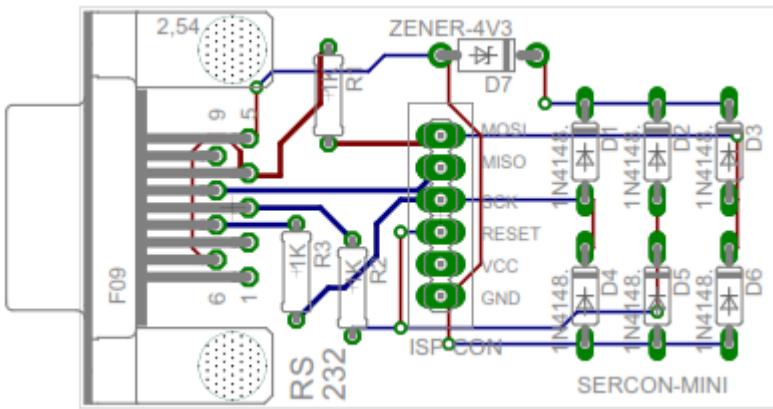


Before

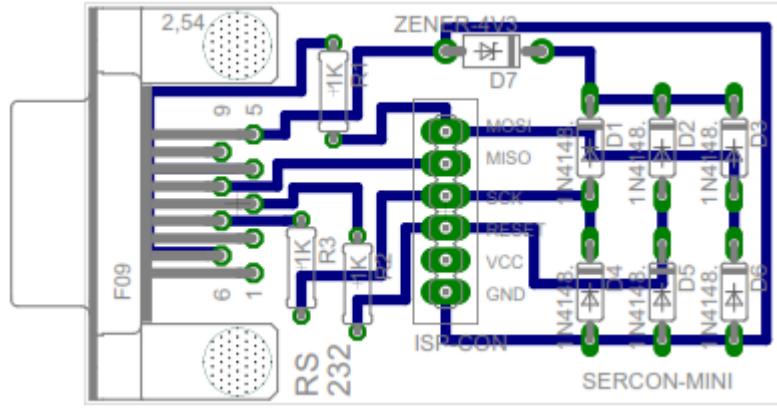


After

Layout routing



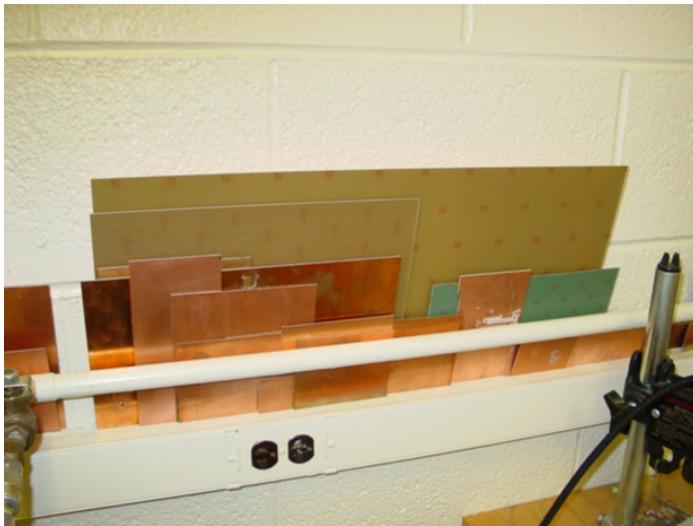
Auto Routing



Manual Routing

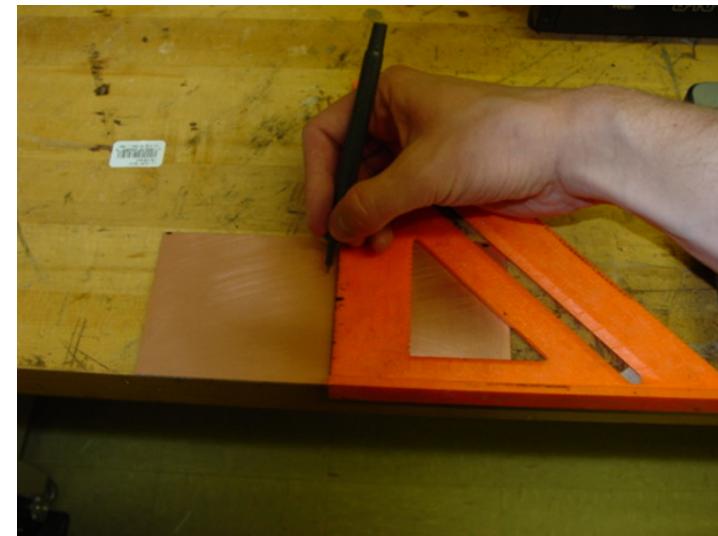
PCB : Fabrication Process

- PCB is made from Copper Clad Board (CCB) i.e type of non-conductive board having copper layer either on one side or both.
- PCB can be one layer PCB or it can be a Multilayer PCB
- Terminology:
 - ***mil***: 1/1000 inch
 - ***thou***: same as mil
 - ***pitch***: Spacing between components (e.g. pins on an IC package)
 - ***Pad***: place around pin of a component where soldering is done
 - ***Trace***: copper track interconnecting components/signal sources
 - ***Drill***: hole through which pins of components are passed and soldered at back
 - ***Silk screen***: top layer of PCB having names in white color
 - ***Reflow/wave soldering***: industrial methods for mass-production soldering



Pick an appropriate copper plated board as per requirement.

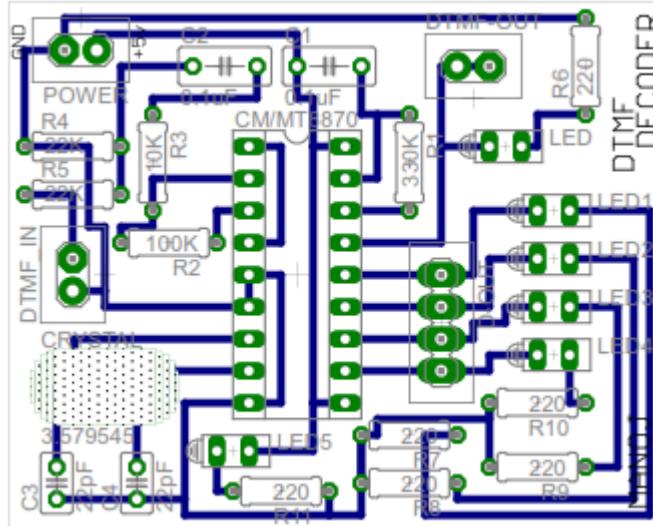
Mark the board to the correct size for your project. ****For best results, add about $\frac{1}{2}$ inch to each dimension!****



Use the metal shear in the shop, or tin snips to cut your board.

Prepare Layout

- Layout is the actual realization of the circuit
- Could be done manually or by the help of any designing software available
- Available software's:- *ORCAD Capture(Cadence), Mentor Graphics, Eagle Layout Editor, Express PCB, PROTEL, Tiny CAD etc...*

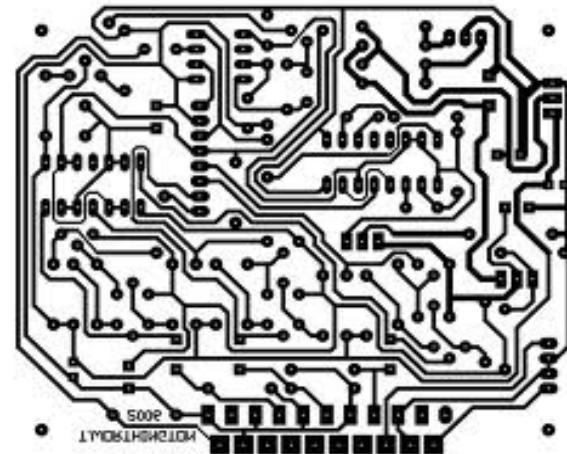


Pads, Traces and silk screen all are clearly visible : Single Layer PTH Board

Negative Development

- Take an undeveloped photofilm
- Align the photofilm with the layout prepared earlier
- This combination is then exposed to light radiations
- Then separate the photofilm and the layout
- Photofilm is dipped into the *negative development solution* for about 1 or 2 min.
- Then the photofilm is dipped in *fixing solution* for 1-2 mins
- Then the photofilm is left to dry
- The negative is obtained

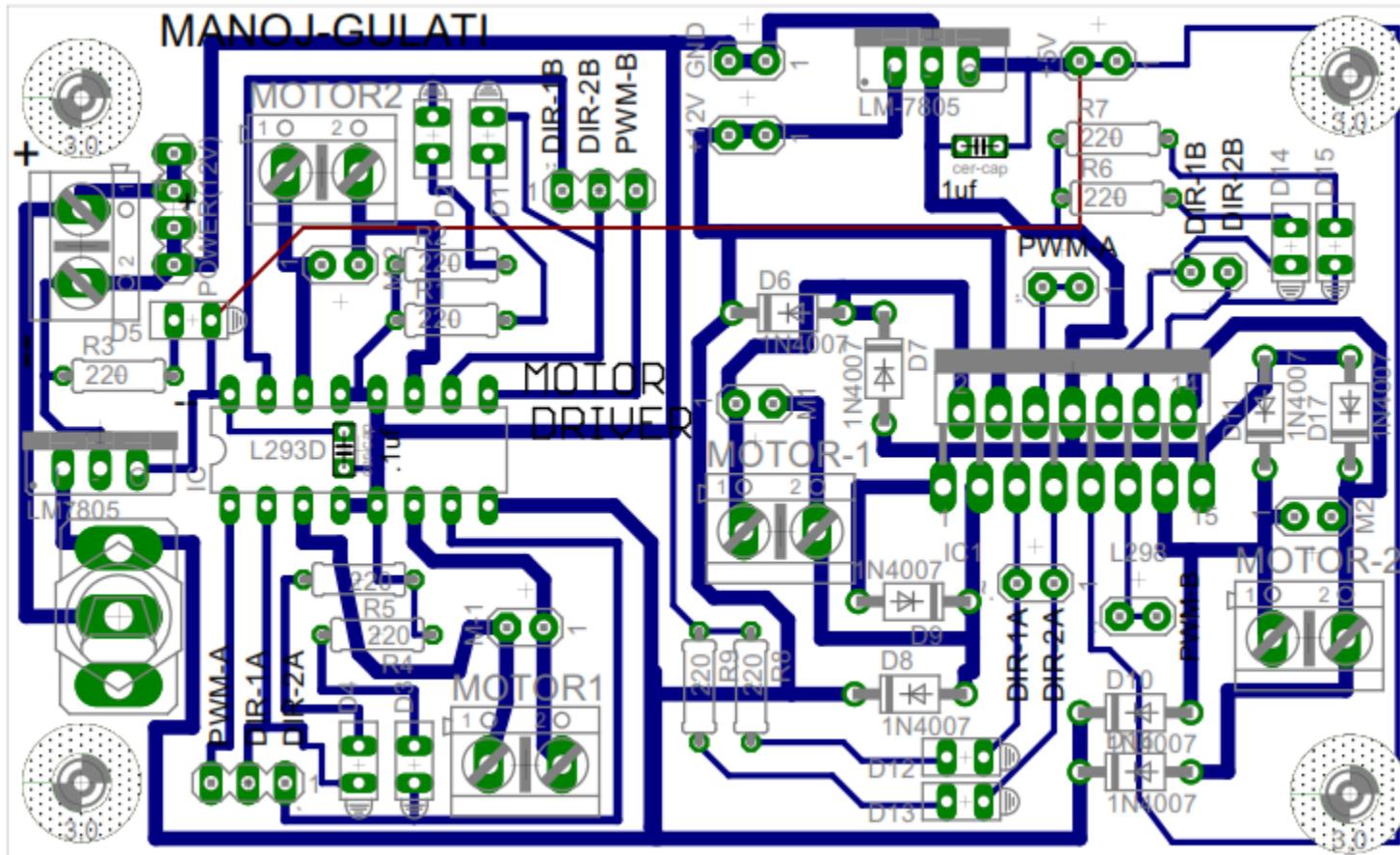
Image Courtesy:
members.optusnet.com.au



Further....

- CCB is taken and dipped in a photoresist solution
- Resultant is then dried with a heater within an oven
- The negative prepared earlier is then aligned with the processed CCB and exposed to ultraviolet radiations
- The photopolymer on the CCB which is exposed to the UV radiation becomes tougher and other part remains softer
- This is resistant to developer solution
- It is then dipped into the Developer solution
- The softer layer of the photopolymer is removed by the solution

Layout : L293/298 Motor Driver



Dying

- The resultant CCB board is then dipped in the dying solution
- It is done for the protection of the track on the circuit board
- The track obtains a specific color
- The part of the board which is covered by dye remains protected from the etching solution, the other part is removed away

Etching

- The resultant board is then dipped into the FeCl_3 solution.
- The arrangement is left undisturbed for etching for 15-20 min depending upon the strength of the solution
- the solution must be hot enough to carry on the process

Home Made PCB

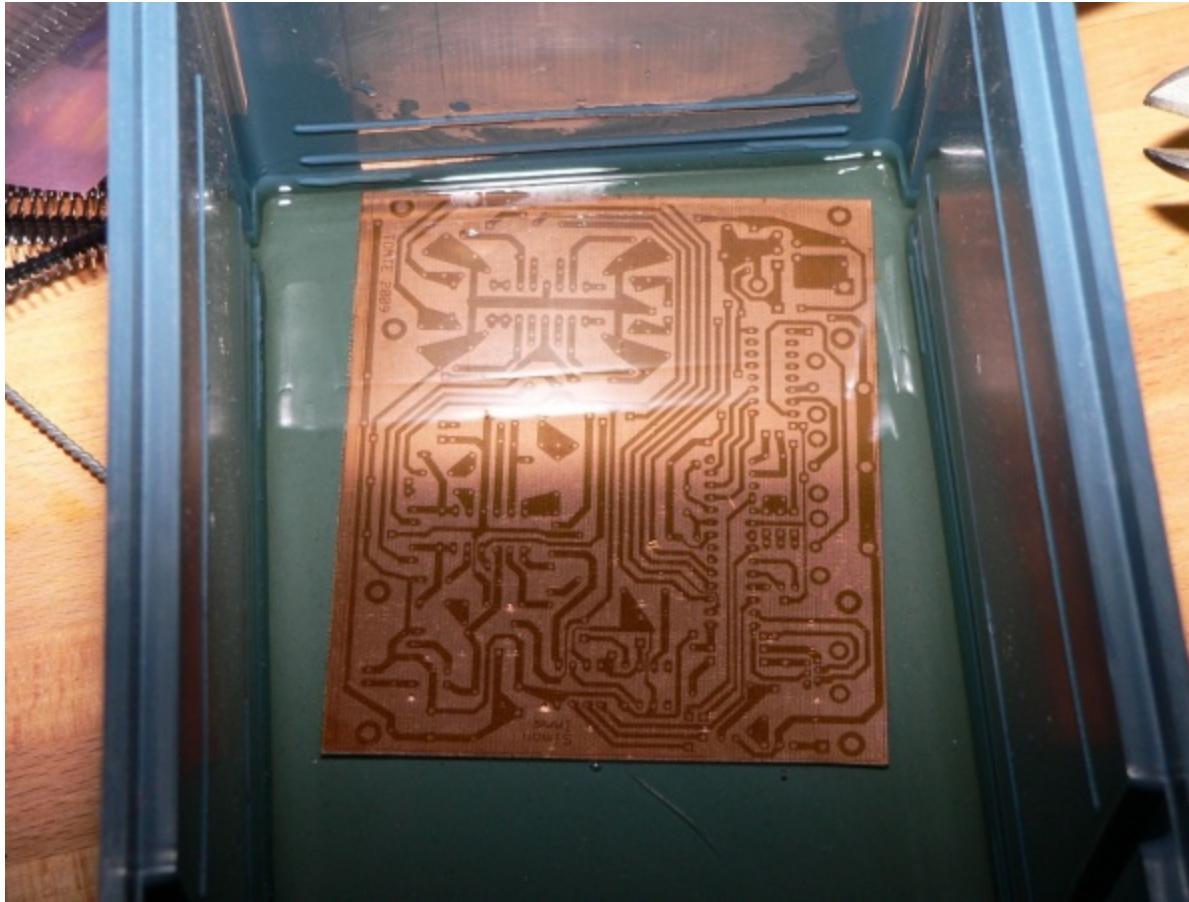
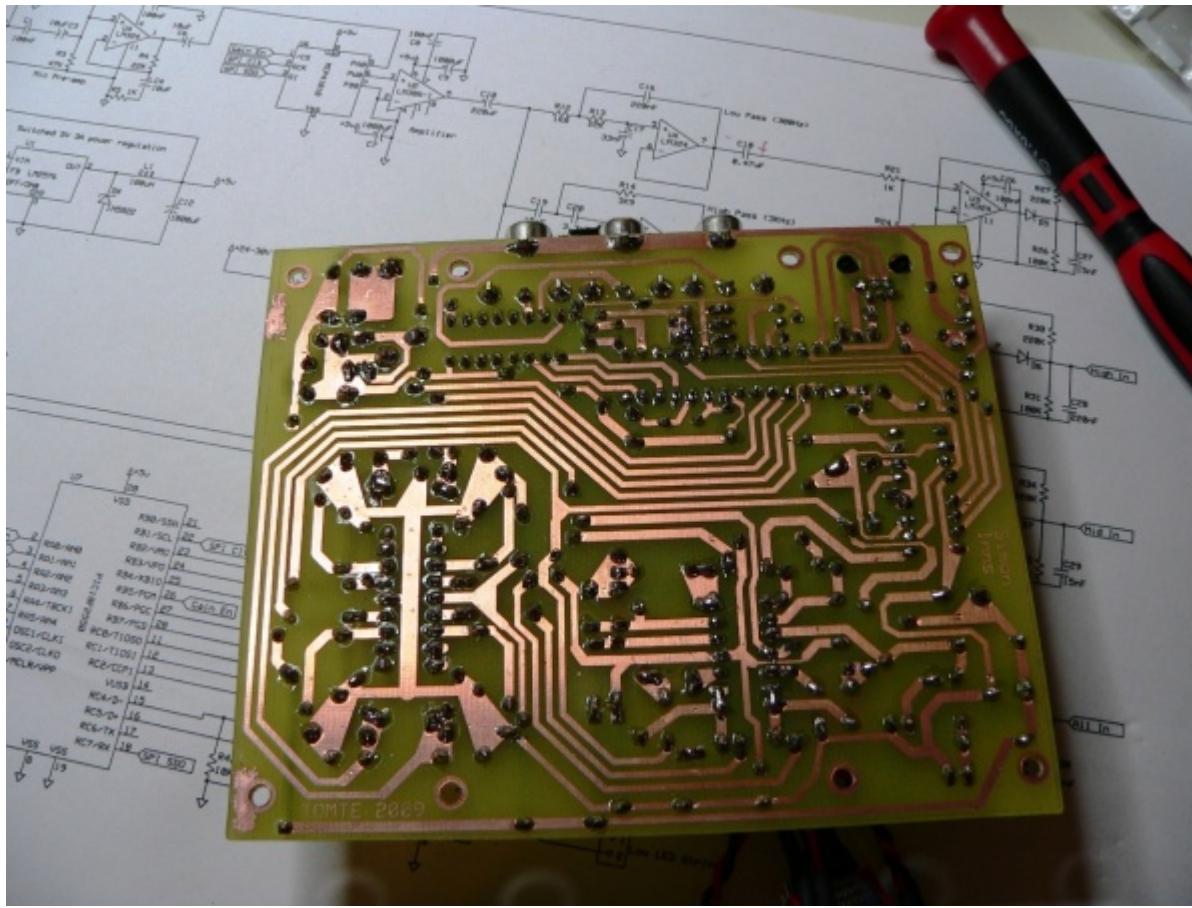


Image Courtesy: http://www.waitingforfriday.com/index.php/LED_Colour_Organ

After component soldering



18/09/15

(20)

Image Courtesy: http://www.waitingforfriday.com/index.php/LED_Colour_Organ

Masking

- In this process we put PCB in to a masking solution which actually provides a protective layer to our tracks and also work as an insulation to external sources.
- Usually it is Green in color but industrial PCB's are usually made with Black/RED mask as it looks decent. But costly too.

Bare PCB after Masking

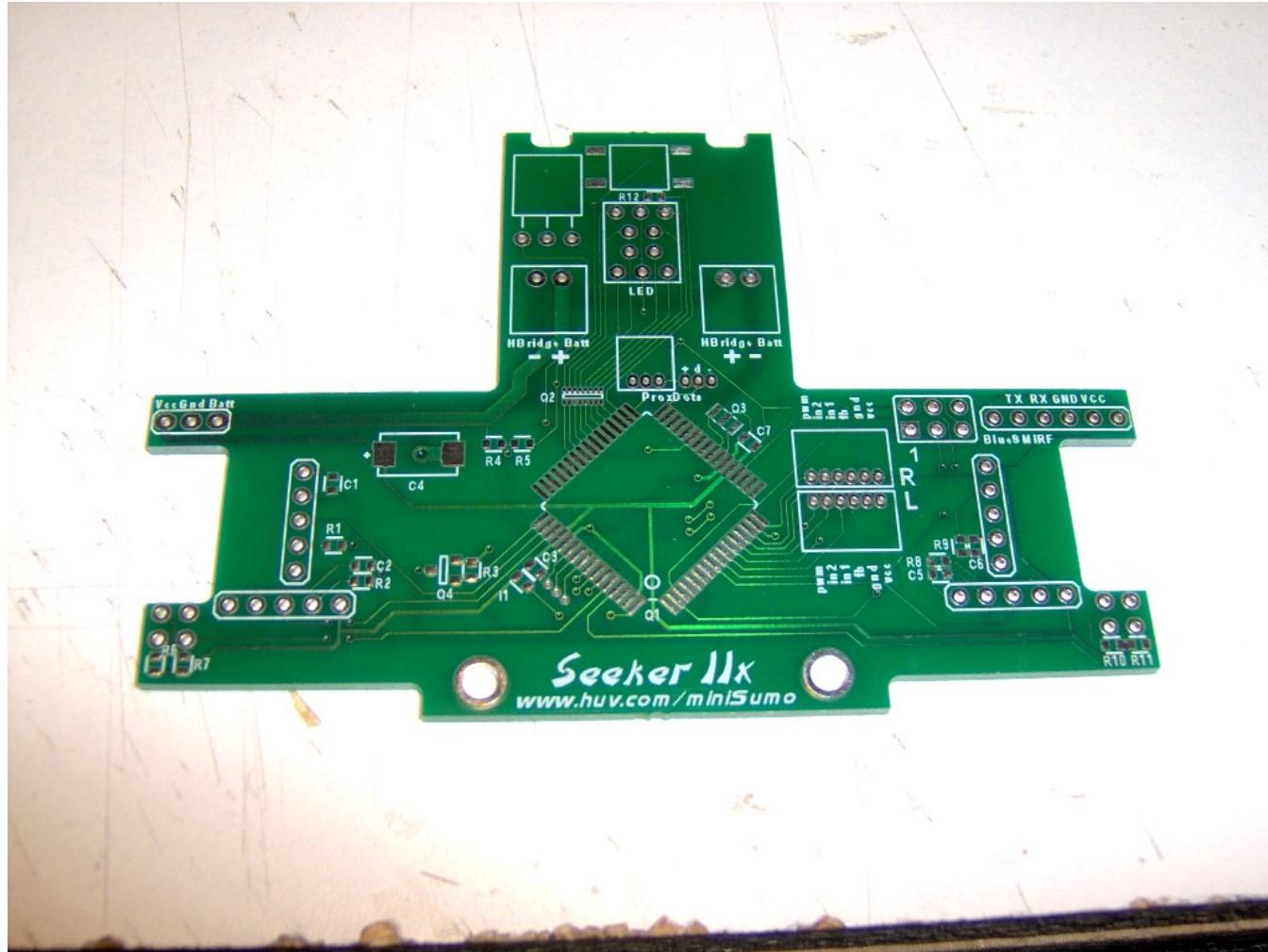


Image Courtesy: <http://www.huv.com/minisumo/seeker2x/>

Tips for making good PCB

- Select components carefully keeping in mind design requirements
- Keep tracks neat
- Use large tracks when possible, no smaller than 10 mils.
- Keep at least 10 mils spacing between tracks
- Add test points
- Use thick pads for High Current flow connections

Thanks for your concentration

18/09/15

You can Ask Queries
er.manojgulati@gmail.com

Now lets Start with the CAD tools

(24)