

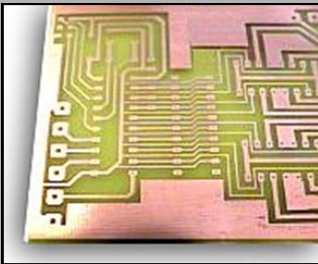
How To Make a PCB



Skill Level: Beginner

Have any of you ever wondered how to make a Printed Circuit Board (PCB)? In this article I will explain the different technics you can use to make your own home mode PCB's.

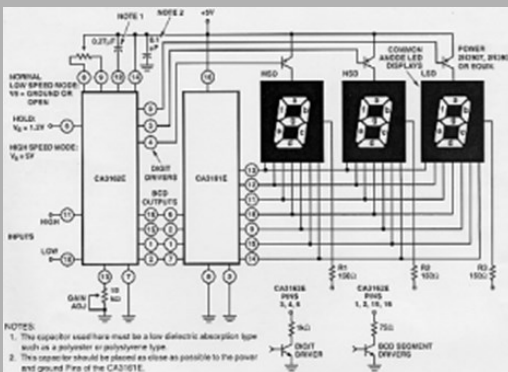
What is a PCB actually



Printed Circuit Board (PCB) is a mechanical assembly consisting of layers of fiberglass sheet laminated with etched copper patterns. It is used to mount electronic parts in a rigid manner suitable for packaging. Also known as a Printed Wiring

Board (PWB).

First things first: The schematic



A schematic diagram must be made that shows the connection of the parts on the board. Each part on the schematic should have a reference designator that matches the one shown on the Bill of Materials (BOM). Many schematic layout programs will allow automatic generation of the BOM like Eagle or Express PCB.

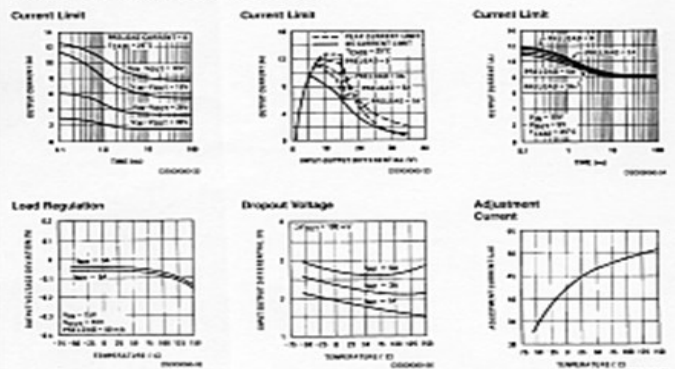
The parts to be mounted on the PCB should be detailed on the parts list. Each part should be identified by a unique reference designator and a part description (i.e. a resistor might be shown as reference designator "R1" with a description of "1/2 Watt Carbon Film resistor resistance value 560 Ohm"). Any additional information useful to the assembly process can be included on this list, such as mounting hardware, part spacers, connector shrouds, or any other material not shown in the schematic diagram.

Choose components from Data Sheets

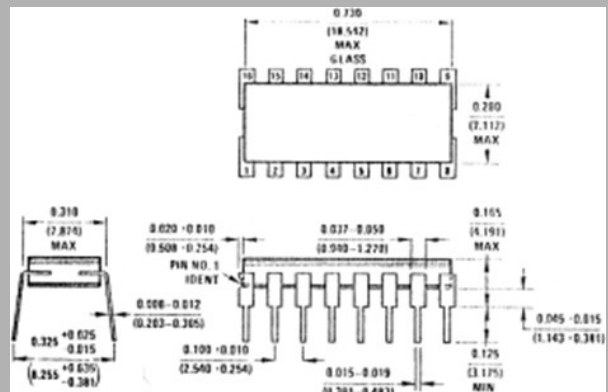
Symbol	Parameter	Conditions	LM338			Units
			Min	Typ	Max	
ΔV_{out}	Temperature Stability	$T_{amb} = 25^\circ C, I_o = I_{max}$	1			%
I_{load}/V_{in}	Minimum Load Current	$V_{in} - V_{out} = 20V$	3.5			10 mA
I_{CL}	Current Limit	$V_{in} - V_{out} = 5.10V$ OC 0.5 ms Peak	5	9	12	A
		$V_{in} - V_{out} = 20V$	7	12	1	A
V_{out}	RMS Output Noise, % of V_{out}	10 Hz \leq f \leq 10 kHz	0.033			%
ΔV_{in}	Ripple Rejection Ratio	$V_{out} = 10V, f = 100 Hz, C_{out} = 0.1 \mu F$	60			dB
		$V_{out} = 10V, f = 100 Hz, C_{out} = 10 \mu F$	75			dB
	Long-Term Stability	$T_j = 125^\circ C, 5000 hrs$	0.3			1 %
θ_{JC}	Thermal Resistance Junction to Case	K Package	7			$^\circ C/W$
		T Package	4			$^\circ C/W$
θ_{JA}	Thermal Resistance Junction to Ambient (Free Heat Sink)	K Package	35			$^\circ C/W$
		T Package	50			$^\circ C/W$

Note 1: Absolute Maximum Ratings indicate limits beyond which damage to the device may occur. Operating Ratings indicate conditions for which the device is intended to be functional, but do not guarantee specific performance limits. For guaranteed specifications and test conditions, see the Electrical Characteristics.
Note 2: These specifications are applicable for power dissipation up to 500mW for the TO-18 (K) package and 250mW for the TO-220 (T) package. Power dissipation is guaranteed at these values up to 10V individual outputs. Above 10V differential, power dissipation will be limited by thermal protection circuitry. All limits (i.e., the numbers in the Min. and Max. columns) are guaranteed by National's MOJL (average Output Quality Level).
Note 3: Regulation is measured at a constant junction temperature, using pulse loading with a low duty cycle. Changes in output voltage due to heating effects are shown under the specifications for thermal regulation.
Note 4: Refer to ECT-0308 drawing for military applications of LM338.

Typical Performance Characteristics



Part manufacturers provide data sheets to be used by the circuit designer to select parts for the circuit. If we are to design the PCB, these sheets should also have the physical dimensions of the part included. Normally you could find datasheets from manufacturers web sites. If each part type to be used on the board does not have a data sheet, you should procure a sample part you can measure to define this data yourself. This measurement method is far less accurate than using the part manufacturer's information, especially if there is a large tolerance on the part, but it is better than just guessing.



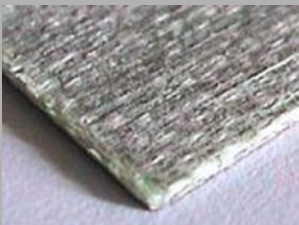
Choose Board type and dimension



A functional PCB is not a finished product. It will always require connections to the outside world to get power, exchange information, or display results. It

will need to fit into a case or slide into a rack to perform its function. There may be areas that will require height restrictions on the board (such as a battery holder molded into the case or rails in a rack the board is supposed to slide into). Tooling holes and keep-out areas may be required in the board for assembly or manufacturing processes. All these outside factors need to be defined before the board can be designed, including the maximum dimensions of the board and the locations of connectors, displays, mounting brackets, or any other external features. The function of a PCB includes the thickness of the copper laminated to the surfaces. The amount of current carried by the board dictates the thickness of this copper foil. Normally the thickness of the copper foil is standard. Also you can choose between different board types for material and number of layers.

Material



Fiberglass-resin laminate (FR4). A rigid PCB of thickness 1.6mm (conventional) or 0.8mm.



As distinct from Fiberglass, Phenolic is a cheaper PCB laminate material. A rigid PCB of thickness 1.6mm (conventional) or 0.8mm.

Layers



One layer of copper. Normally the wire-leaded components must be mounted on only one side of the PCB, with all the

leads through holes, soldered and clipped. You can also mount the components on the track surface using Surface Mount Devices (SMD). Surface mount circuitry is generally smaller than conventional. In practice, most boards are a mix of surface mount and conventional components. This can have its disadvantages as the two technologies require different methods of insertion and soldering. Conventional circuitry is generally easier to debug and repair.



Two layers of copper, one on each side of the board. The components must be mounted on only one side of the PCB but you can also mount components on both sides of the PCB. Normally only surface mount circuitry would be mounted on both sides of a PCB. The components must be mounted using both through-holes technology or Surface Mount Technology (SMT) or Surface Mount devices (SMD).



A PCB Laminate may be manufactured with more than two layers of copper tracks by using a sandwich construction. The cost of the laminate reflects the number of layers. The extra layers may be used to route more complicated circuitry, and/or distribute the power supply more effectively.

Layout

The PCB layout can be drawn either manually or by CAD (Computer Aid Design) software. The manual process is useful and quick only for very easy PCBs, for more complex PCBs I suggest the second way. These days inexpensive computer software can handle all aspects of a PCB's pre-processing. A few general rules of thumb that can be used when laying out PC boards are:

- Generally, it is best to place parts only on the top side of the board. First place all the components that need to be in specific locations. This includes connectors, switches, LEDs, mounting holes, heat sinks or any other item that mounts to an external location. Give careful thought when placing component to minimize trace lengths. Put parts next to each other that connect to each other. Doing a good job here will make laying the traces much easier.
- Arrange ICs in only one or two orientations: up and down, or, right and left. Align each IC so that pin one is in the same place for each orientation, usually on the top or left sides. Position polarized parts (i.e. diodes, and electrolytic caps) with the positive leads all having the same orientation. Also use a square pad to mark the positive leads of these components. You will save a lot of time by leaving generous space between ICs for traces. Frequently the beginner runs out of room when routing traces. Leave 9mm - 15mm between ICs, for large ICs allow even more space.
- Parts not found in the component library can be made by placing a series of individual pads and then grouping them together

- When choosing a pad and hole size for the pins of component, keep in mind that the hole size refers to the drill size used when the board is made. After the hole is plated, the diameter will shrink by 0.127mm - 0.1778mm. From the pin diameter, add 0.2mm or more when selecting a hole size. After placing all the components, print out a copy of the layout. Place each component on top of the layout. Check to insure that you have allowed enough space for every part to rest without touching each other.

Placing Power and Ground Traces

After the components are placed, the next step is to lay the power and ground traces. It is essential when working with ICs to have solid power and ground lines, using wide traces that connect to common rails for each supply. It is very important to avoid snaking or daisy chaining the power lines from part-to-part. One common configuration is to use the bottom layer of the PC board for both the power and ground traces. A power rail can be run along the front edge of the board and a ground rail along the rear edge. From these rails attach traces that run in between the ICs. The ground rail should be very wide, perhaps 2.54mm and all the other supply lines must be 1.27mm or wider. When using this configuration, the remainder of the bottom layer is then reserved for the vertical signal traces. Or a ground plain can be used. This is when one or both sides of the PCB is used for ground with the traces running in between the ground plain. This configuration helps a lot to dampen capacitive crosstalk in the traces and also provides a good solid ground plain that again helps prevent ground bounce.

Placing Signal Traces

The process to connect the parts together is known as routing and can be done manually or automatically. If you use auto route software it's a good idea to route critical signals by hand. This allows the signal to be routed with less bends and vias than if the auto router does it. Some signals may require special treatment such as grounding or specific lengths that may be easier to do before auto routing than after. These traces should be flagged as fixed so the auto router doesn't move them. When placing traces, it is always a good practice to make them as short and direct as possible. Use vias to move signals from one layer to the other. A via is a pad with a plated-through hole. Generally, the best strategy is to layout a board with vertical traces on one side and horizontal traces on the other. Add vias where needed to connect a horizontal trace to a vertical trace on the opposite side. A good trace width for low current digital and analog signals is 0.4mm.

Traces that carry significant current should be wider than signal traces. The table below gives rough guidelines of how wide to make a trace for a given amount of current.

Temp Rise	10°C	20°C	30°C
Trace Width in Mil	Maximum Current in Amps		
10	1.0	1.2	1.5
15	1.2	1.3	1.6
20	1.3	1.7	12.4
25	1.7	2.2	2.8
30	1.9	2.5	3.2
50	2.6	3.6	4.4
75	3.5	4.5	6.0
100	4.2	6.0	7.5
200	7.0	10.0	13.0
250	8.3	12.3	15.0

Please do note that Mil and mm is not the same, 10 mil is plus minus 0.4mm.

When placing a trace, it is very important to think about the space between the trace and any adjacent traces or pads. You want to make sure that there is a minimum gap of 0.18mm between traces, 0.254mm is better. Leaving less blank space runs the risk of a short developing in the board manufacturing process. It is also necessary to leave larger gaps when working with high voltage.

It is a common practice to restrict the direction that traces run to, horizontal, vertical, or at 45 degree angles. When placing narrow traces, 0.635mm or less, avoid sharp right angle turns. The problem here is that in the board manufacturing process, the outside corner is often etched too narrow. The solution is to use two 45 degree bends with a short leg in between.

It is good idea to place text on the top layer of your board, such as a product or company name. Text on the top layer can be helpful to insure that there is no confusion as to which layer is which when the board is manufactured.

Checking Your Work

After all the traces are placed, it is best to double check the routing of every signal to verify that nothing is missing or incorrectly wired. Do this by running through your schematic, one wire at a time. Carefully follow the path of each trace on your PC layout to verify that it is the same as on your schematic. After each trace is confirmed, mark that signal on the schematic with a highlighter. Inspect your layout, both top and bottom, to insure that the gap between

every item (pad to pad, pad to trace, trace to trace) is 0.18mm or greater. Check for missing vias. An easy way to check for missing via is to first print the top layer, then print the bottom. Visually inspect each side for traces that don't connect to anything. When a missing via is found, insert one. Check for traces that cross each other. This is easily done by inspecting a printout of each layer. Metal components such as heat sinks, crystals, switches, batteries and connectors can cause shorts if they are placed over traces on the top layer. Inspect for these shorts by placing all the metal components on a printout of the top layer. Then look for traces that run below the metal components.

Different methods to make PCBs

Etching is probably the easiest and most cost effective.

Etching is the process of chemically removing the unwanted copper from a plated board. You must put a mask or resist on the portions of the copper that you want to remain after the etch. These portions that remain on the board are the traces that carry electrical current between devices.



It's messy stuff but easier to get and cheaper than most alternatives. It attacks ANY metal including stainless steel, so when setting up a PCB etching area, use a plastic or ceramic sink, with plastic fittings & screws wherever possible, and seal any metal screws etc. with silicone. If copper water pipes may get splashed or dripped, sleeve or cover them in plastic.

Fume extraction is not normally required, although a cover over the tank or tray when not in use is a good idea. You should always use the hexahydrate type of ferric chloride, which is light yellow, and comes as powder or granules, which should be dissolved in warm water until no more will dissolve. Adding a teaspoon of table salt helps to make the etchant clearer for easier inspection.

Anhydrous ferric chloride is sometimes encountered, which is a green-brown powder. Avoid this stuff if at all possible, use extreme caution, as it creates a lot of heat when dissolved - always add the powder very slowly to water, do not add water to the powder, and use gloves and safety glasses.

You may find that solution made from anhydrous FeCl doesn't etch at all, if so, you need to add a small amount of hydrochloric acid and leave it for a day or two. Always take extreme care to avoid splashing when dissolving either type of FeCl - it tends to clump together and you often get big chunks coming out of the container & splashing into the solution. It will damage eyes and permanently stain cloth-

ing and pretty much anything else - use gloves and safety glasses and wash off any skin splashes immediately.

If you're making PCBs in a professional environment, where time is money, you really should get a heated bubble-etch tank. With fresh hot ferric chloride, a PCB will etch in well under 5 minutes, compared to up to an hour without heat or agitation. Fast etching also produces better edge quality and consistent line widths.

If you aren't using a bubble tank, you need to agitate frequently to ensure even etching. Warm the etchant by putting the etching tray inside a larger tray filled with boiling water - you want the etchant to be at least 30-60°C for sensible etch times.



Sodium Persulphate Etchant is supplied as a dry, easily mixed etchant for copper clad printed circuit boards and other copper bearing materials. When used in conjunction with the catalyst, a very constant etch rate can be maintained throughout the life of the bath. After mixing, the bath has a practical life of about three (3) weeks and a copper capacity of approximately 29.96 - 37.45 grams / liter of etchant.

Sodium Persulphate Etchant, when compared to Ferric Chloride and other copper etchants, has several distinct advantages.

Cleanliness: Sodium Persulphate will not stain clothes, skin or tanks.

Rinsing: Sodium Persulphate rinses easily and leaves no residue in plain water.

Etching speed: Sodium Persulphate attains a fast etching speed and with regular additions of the catalyst, maintains a relatively even etching rate throughout its entire mix life. However, as with all etchants, Sodium Persulphate has several disadvantages:

Short active life: once Sodium Persulphate is mixed, it has a tank life of three (3) weeks, maximum, whether or not it is used.

Aggressiveness: Sodium Persulphate will attack natural fibers such as cotton, wool and linen. So you won't end up with a stain on your cloths but with a hole if you don't rinse of the Etchant before it dries.

The Art Work

Manually (direct draw)

One way to put a pattern on the board is the direct draw approach. Using either a resist pen to draw your circuit, or by using specialty tapes (dry transfers), donuts and the lot, you layout your circuit traces directly onto the copper surface of the board. The pen technique relies on the waterproof nature of the ink and the tapes as an impervious plastic, both of which prevent the etchant from getting at the copper beneath, hence, all copper is etched away except for where the pattern has been drawn.

This is the quickest way to get a circuit pattern on the board, but it is difficult to position the traces accurately, especially if you are using any IC packages in your design. Plus, since the ink doesn't apply uniformly, there is a risk that the traces will be etched away since the etchant can get to the copper through an extremely thin layer of resist. If you make a mistake you have to start all over again! For these reason, you can use this method only to make very easy low-definition PCBs or to retouch a bit your PCB before etching.

Photographic

I suggest this method to producing consistently high quality PCBs quickly and efficiently, particularly for professional prototyping of production boards.

In this method, a board is covered with a resist material that sets up when exposed to Ultra Violet light. To make a board this way, you must make a positive UV translucent artwork film of your layout pattern which is opaque where you want a circuit trace, and clear where you don't want a trace. After the photo positive film is made from your artwork, it is placed onto the photo sensitized board, and is exposed to the UV. The UV light transmits through the clear portions of the film and cures the photoresist. After that, the board is submerged into a developer bath that develops and remove the sensitized photoresist. The resist that is left is in the shape of the artwork that represents your circuit. The advantage to this approach is accurate and neat traces, and once you make the artwork film, it can be used over and over to make additional boards.

You'll never get a good board without good artwork, so it is important to get the best possible quality at this stage. The most important thing is to get a clear sharp image with a very solid opaque black. These days, artwork will almost always be drawn using either a dedicated PCB CAD program, or a suitable drawing / graphics package.

The artwork must be printed such that the printed side will be in contact with the PCB surface when exposing, to avoid blurred edges. In practice this means that if you design the board as seen from the component side, the bottom (solder side) layer should be printed the 'correct' way round, and the top side of a double-sided board must be printed mirrored. Artwork quality is very dependent on both the output device and the medium used.

Photoresist PCB laminates

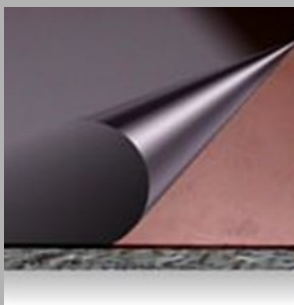


Spray photo sensitive resist. It's very hard to use, as you always get dust settling on the wet resist. I wouldn't recommend it unless you have access to a very clean and ventilated area or drying oven, or only want to make low-resolution PCBs. In any case, to use the positive photoresist spray you must:

- 1) Make a lot of practice boards.
- 2) Cleaning: Degrease the surface before application of product.
- 3) Application of the coating: Spray the PCB board in a subdued light from a distance of approximately 20 cm until the coating is visible.

This work must be carried out in absolutely dust-free conditions. Then the coating must be dried (20°C = 24 hour, better 70°C = 15 min).

Note that the photoresist sprays are normally EXTREMELY FLAMMABLE.



Pre-coated photoresist fiberglass (FR4) board. Always use good quality pre-coated photoresist fiberglass (FR4) boards. Check carefully for scratches in the protective covering, and on the surface after peeling off the covering. You don't need darkroom or subdued lighting when handling

boards, as long as you avoid direct sunlight, minimize unnecessary exposure, and develop immediately after UV exposure.

Exposure



The photoresist board needs to be exposed to ultra-violet light through the artwork, using a UV exposure box or any other UV source, CFL bulbs also work nice.

UV exposure units can easily be made using standard fluorescent lamp ballasts and UV tubes. For small PCBs, two or four 8 watt 12" tubes will be adequate, for larger (A3) units, four 15" 15 watt tubes are ideal.

To determine the tube to glass spacing, place a sheet of tracing paper on the glass and adjust the distance to get the most even light level over the surface of the paper. Even illumination is a lot easier to obtain with a 4-tube units. The UV tubes you need are those sold either as replacements for UV exposure units, insect killers or 'black light' tubes for disco lighting etc. They look white or occasionally black/blue when off, and light up with a light purple, which makes fluorescent paper glow brightly. DO NOT use short-wave UV lamps like EPROM eraser tubes or germicidal lamps, which have clear glass - these emit short-wave UV which can cause eye and skin damage.

A timer which switches off the UV lamps automatically is essential, and should allow exposure times from 2 to 20 minutes. It is useful if the timer has an audible indication when the timing period has completed. A timer from a scrap microwave oven would be ideal.

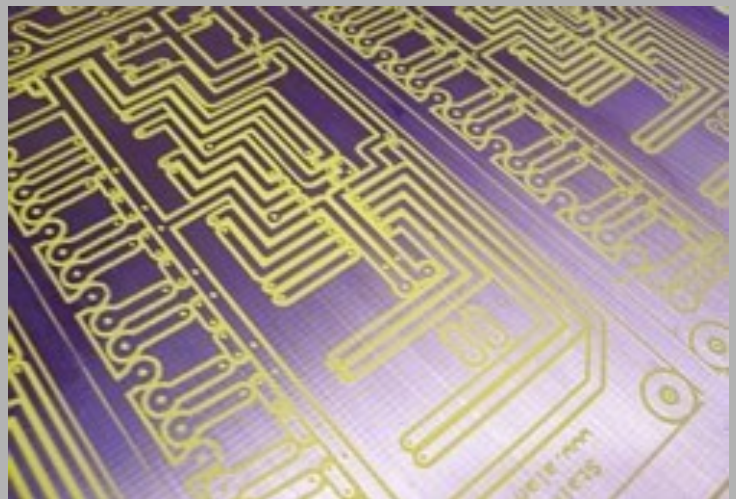
Short-term eye exposure to the correct type of UV lamp is not harmful, but can cause discomfort, especially with bigger units. Use glass sheet rather than plastic for the top of the UV unit, as it will flex less and be less prone to scratches. If you do a lot of double-sided PCBs, it may be worth making a double-sided exposure unit, where the PCB can be sandwiched between two light sources to expose both sides simultaneously.

You will need to experiment to find the required exposure time for a particular UV unit and laminate type, expose a test piece in 30 second increments from 2 to 20 minutes, develop and use the time which gave the best image. Generally speaking, overexposure is better than underexposure.

For a single-sided PCB, place the artwork with toner side up on the UV box glass, peel off the protective film from the laminate and place it sensitive side down on top of the artwork. The laminate must be pressed firmly down to ensure good contact all over the artwork, and this can be done either by placing weights on the back of the laminate, or by fitting the UV box with a hinged lid lined with foam rubber, which can be used to clamp the PCB and artwork.

To expose double-sided PCBs, print the solder side artwork as normal, and the component side mirrored. Place the two sheets together with the toner sides facing, and carefully line them up, checking all over the board area for correct alignment, using the holes in the pads as a guide. A light box is very handy here, but it can be done with daylight by holding the sheets on the surface of a window. If printing errors have caused slight miss-registration, align the sheets to 'average' the error across the whole PCB, to avoid breaking tracks when drilling.

When they are correctly aligned, staple the sheets together on two opposite sides (3 sides for big PCBs), about 10mm from the edge of the board, forming a sleeve or envelope. The gap between the board edge and staples is important to stop the paper distorting at the edge. Use the smallest stapler you can find, so the thickness of the staple is not much more than that of the PCB. If you do not have a double-sided exposure unit, expose each side in turn, covering up the top side with a reasonably light-proof soft cover when exposing the underside (rubber mouse mats are ideal for this). Be very careful when turning the board over, to avoid the laminate slipping inside the artwork envelope and ruining the alignment. After exposure, you can usually see a faint image of the pattern in the photosensitive layer.



After development

Developing

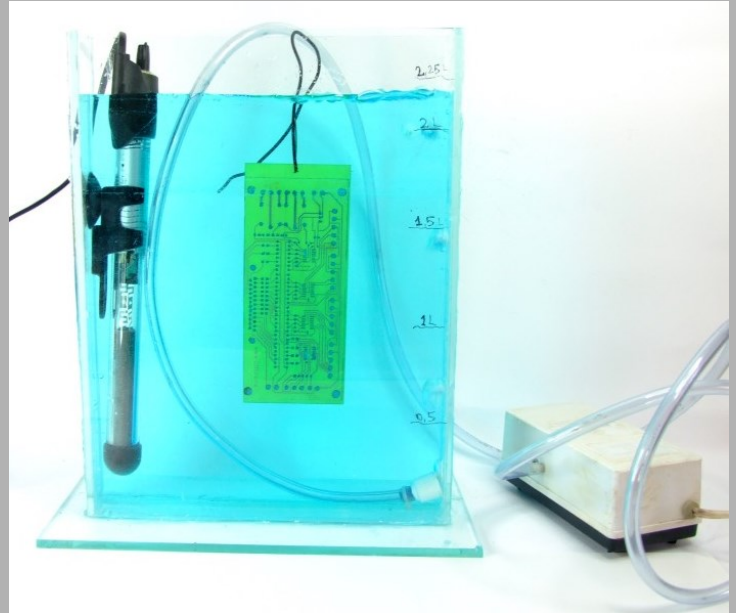
VN447 Etching tank from Electronics 123



Silicate based products comes as a liquid concentrate. This stuff has huge advantages over sodium hydroxide, most importantly is very hard to over-develop. You can leave the board in for several times the normal developing time without noticeable degradation. This also means it's not temperature critical, no risk of stripping at warmer temperatures. Made-up solution also has a very long shelf-life, and lasts until it's used up, the concentrate lasts for at least a couple of years.

The lack of over-developing problems allows you to make the solution up really strong for very fast developing. The recommended mix is 1 part developer to 9 parts water, but you can make it stronger to develop in few seconds without the risk of over-development damage. You can check for correct development by dipping the board in the ferric chloride very briefly, the exposed copper should turn dull pink almost instantly. If any shiny copper colored areas remain, rinse and develop for a few more seconds. If the board was under-exposed, you can get a thin layer of resist which isn't removed by the developer. You can often remove this by gently wiping with a dry paper towel, which is just abrasive enough to remove the film without damaging the pattern.

You can either use a photographic developing tray or a vertical tank for developing, a tray makes it easier to see the progress of the development. You don't need a heated tray or tank unless the solution is really cold ($<15^{\circ}\text{C}$).



Product code: VN447

Cleaning



Use acetone or alcohol to remove resist. Clean copper board with steel wool. Rinse cleaned board with soap and water. Be sure to remove all soap residue. Dry thoroughly with lint-free cloth. Be sure to scrape any burrs that appear on the edge of the board that may have resulted from the cutting/shearing process.

Electronics 123

Contact us on :
Tel: 012 3322356
Fax: 012 3320487

Visit our website :
<http://www.electronics123.co.za>

Email us :
Electronics123