How I Make PC Boards

(One of many methods)

by: Tom Hammond, NØSS
3 March 2006

Too many years ago (30 or more), I decided to make a PC board. I had no experience at all in designing PC boards, and other than assembling Heathkits, I had little more experience in building circuits. But I decided it was time I learned.

A local ham, KØETY, showed me how to use 10 square to the inch graph paper to lay out the circuit design, based upon the schematic. He also showed me how to lay my hand-drawn design over a piece of clean copper-clad board and use a nail to punch the points where each through-hole was to be drilled for a component lead. He then showed me how to apply (my XYL's) nail polish to the copper-clad board in order to draw in the traces (lands) connecting each of the marked holes to be drilled. Once dried, the soon-to-be PC board was immersed in ferric chloride (etchant) and after a period of time, the copper which was not covered by the nail polish would be chemically dissolved and 'etched' away, leaving the desired pattern on the board. My first PC board... crude, but it still worked.

As time went on, I graduated from painting-on nail polish to using waterproof marking pens (fresh Sharpies) to draw the pads and lines onto the copper-clad board. If I did a good job of drawing, and made sure that the waterproof ink was applied heavily enough to the board, the etchant would not eat through the ink (hereafter referred to as "resist") and I would wind up with a still crude, but cleaner looking PC board.

The big problem was that this process was all manual, and I could only create one-at-a-time copies, with a fairly significant amount of effort required to even create that single board. By this time, I had gotten into creating 'kits' which were used to generate 'building' interest among members of our local radio club. These kits required that multiple copies of the same board be produced, and that they all looked the same. I had to find a better way, and a method which would yield an less 'amateurish' looking product.

About that time, I came across a copy of Corel Draw, a vector graphics drawing program, which I found to be relatively easy to learn. I found that Corel Draw allowed me to produce nice, clean PC board designs, and to be able to change the drawing I made to accommodate circuit changes much more easily than I'd been able to do with the hand-drawn graph paper designs I was previously using.

My problem then was that I had to come up with a method of using the board designs I had previously drawn to make PC boards.

I soon found that many professional PC board manufacturers were using a method known as 'photo etching'. Photo etching used a piece of copper-clad PC board which had its copper side coated with a light-sensitive emulsion (so it became photo-sensitive to ultraviolet [UV] light). A PC board layout could then be photographically copied to a photo negative film, placed in direct contact with the photo-sensitized PC board and when exposed to UV light, the PC board image would be transferred to the photo-sensitive emulsion on the PC board. A low-budget PC board exposure frame is illustrated in Figure 1.

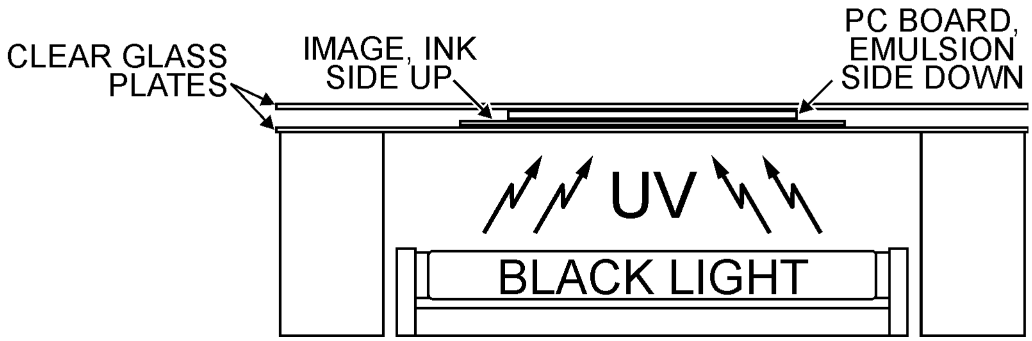


Figure 1 - Budget UV Light Box for Exposing PC Boards

There are at least two types of photo-emulsion resist available for making PC boards:

1. **Positive-working resist** - This resist requires a photo-positive image (e.g. black where copper is to be, and clear where no copper is to be). With this type of resist, the part of the resist which is exposed to the UV light becomes soft and is dissolved when the PC board is later placed into the 'developer' (to remove the softened resist, prior to the board being etched).

2. **Negative-working resist** - This requires a photo-negative image (e.g. clear where copper is to be, and black where no copper is to be). With this type of resist, the part of the resist which is exposed to the UV light becomes solid and is NOT dissolved when the PC board is later placed into the 'developer'. The part of the resist which is not exposed to UV remains soft and will be dissolved by the developer and washed away in a later step.

The company from which I buy my PC board supplies, sells PC board which is pre-sensitized with Kodak KPR-4, a negative-working resist. As a result, the board designs I create require me to produce a photo-negative image for the exposure process.

Using Corel Draw (currently v9), I draw out my initial PC board layout. Several years ago, I took the time to pre-draw a number of commonly-used component outlines and the PC board pad designs which fit those components. This allows me to merely pick up a component (and it's PC board pads) and to place it onto my board layout in the position I wish to eventually have it located. I can always move the component and pads later if I decide I want it elsewhere on the board. A partial PC board layout is shown below (Figure 2) to illustrate this part of the board design process. Note that I work with component and pad images which are TWO TIMES normal size. This provides me with an image which is a bit easier to edit and one which I can then reduce by 50% when I'm ready to actually create the final image to be used when making the PC board.

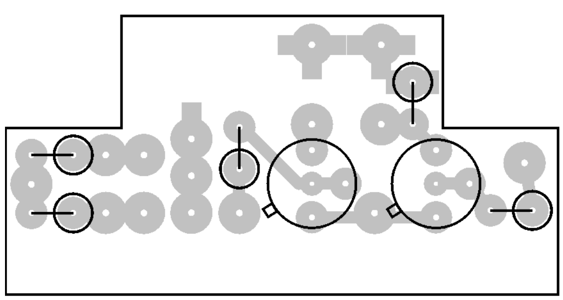


Figure 2. Initial PC board component & Pad Layout

In Figure 2, you will note some lands which come out from a round pad, but which don't connect to anything. These are ground lands which will eventually connect to a large area of copper which will surround the completed layout. You may also notice that there are four holes shown for the two transistor outlines. This is because the transistor specified for this particular board design is a 2N2222 which also comes in a plastic (PN2222) design. The 2N2222 lead configuration is in a triangle, while the PN2222 leads are in a straight line. The extra hole allows either transistor to be used without having to bend leads one direction or the other to fit the holes.

In Figure 2, I started with an outline of my final PC board shape because I knew the dimensions of the enclosure it had to fit (a DB-25 case). Many times, I don't draw the actual board outline until I have completely drawn the component and pad design and compressed it as much as desired. Then, I'll draw the outline of the board, usually allowing some extra length and width to accommodate mounting hardware (screws & nuts, etc.).

Being a basically frugal person (my kids say I'm ”cheap"), I don't wish to deplete my etchant any more rapidly than necessary. As a result, I design my PC boards to remove no more copper than absolutely necessary. This requires me to design areas of no copper around each of the component pads and the connecting copper lands.

Once I have satisfied myself that I have the basic component and pad layout down and in the format I wish to use, I will begin to add copper back to my design. I do this by making identical copies of each pad (or group of pads), placing them BEHIND the copper pads (shown in gray in the illustrations), making them white and using a much wider line width, so they create each pad and land as an island of their own. This is illustrated below in Figure 3. Note that I have 'flooded' the board outline with copper, so you can easily see the white 'cutouts' around each pad and land.

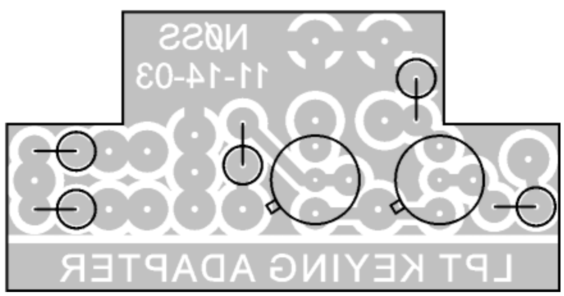


Figure 3 - Completed PC board layout, including ground foil

You will note that the lettering on the PC board is backwards. This is because, when the layout is eventually printed to acetate film, using a laser printer, the ink (toner) will be deposited onto the front side of the film which will then be inverted and placed directly AGAINST the photo-sensitive emulsion of the PC board (for a 'contact' print). This causes the lettering to be correctly printed onto the PC board. However, IF you were going to use this design to make a true PHOTOGRAPHIC negative, you would invert the image before you made the negative since photographic negatives have their emulsion on the BACK side of the film instead of the front side.

When the board has been 'flooded' with ground copper, I double-check (usually triple-check) the layout to ensure that I made no design errors. When I design PC boards, I find that it is very easy for me to miss a design error... probably because I'm too 'close' to the design, and I'm certain I couldn't have made a mistake... so I often ask a disinterested third party to check my layout. This usually results in a clean design.

The next step is to add component nomenclature so I can confirm that I have actually designed-in all the required components. It's very embarrassing to find you left out a crucial component after you've already made several dozen new PC boards! The completed board layout and nomenclature is shown below, in Figure 4.

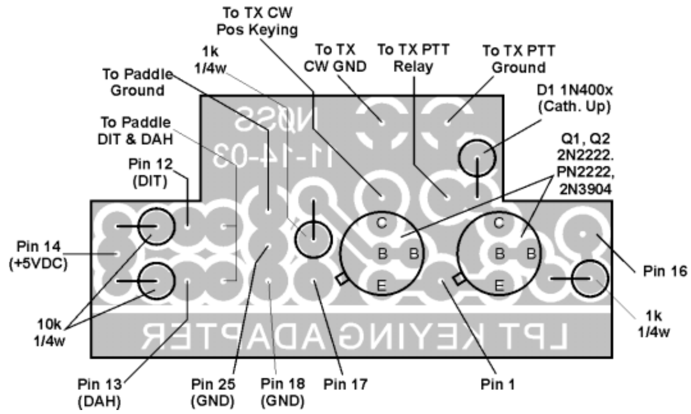


Figure 4 - Completes PC board design, complete with nomenclature

Fully confident that I have forgotten nothing, I now make a copy of the entire PC board layout, reduce it by 50%, to get it down to the actual board size, and I'll change the color of the pads and lands to match whether I'll be using a positive-working or a negative-working resist. See Figure 5, below.

Figure 5. - Positive-working & Negative-working
Resist Layout Resist Layout

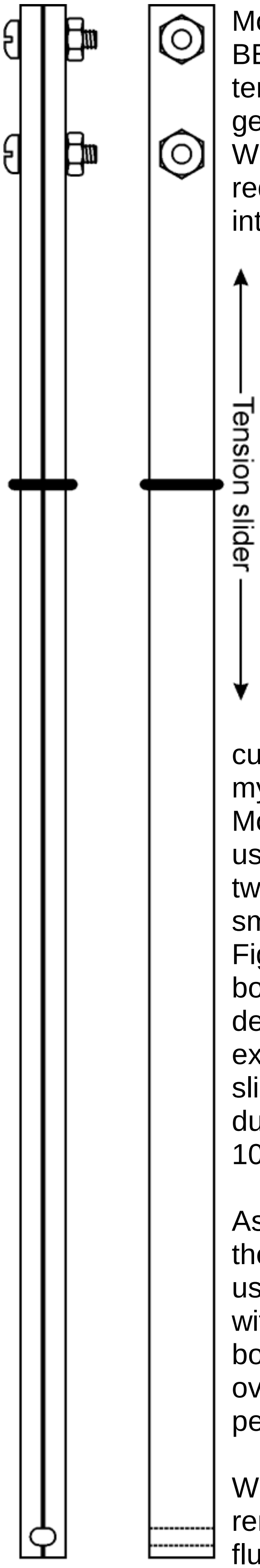
If I'm making single copies of a PC board, I'll print one of these two designs to acetate (transparency) film and proceed with making a contact print onto the PC board. If however I'm making a multi-board run, I'll print multiple copies of the design to the transparency film and I'll expose a number of boards (onto a larger piece of unexposed PC board) at a time.

After the PC board has been exposed, it is placed into a PLASTIC or GLASS (preferably PYREX) dish of developer. Again, the type of developer depends upon the type of sensitizer (resist) used on the PC board itself... you purchase the developer to match the type of sensitizer. Once the board has 'soaked' for the required length of time (usually 3-4 minutes), it is held under running water to remove the developer (and the softened resist) and to reveal the image now printed onto the PC board. Some resists are tinted so you can easily see the printed image, others are not and you must hold the PC board at an angle against the light in order to get an idea of how well the image printed.

ETCHING THE PC BOARD

There are two types of PC board etchant commonly used by hobbyists: Ferric Chloride and Sodium Persulfate. Each of these etchants has its good and bad points. I'll go into a few (but not all) of them at the end of this document.

ALWAYS use a plastic or glass (preferably Pyrex) dish when etching PC boards...!



Most etchants will etch more quickly if warmed a bit. BUT BE CAREFUL if you do warm these chemicals. Keep the temperature to a level which you can easily handle, generally 120° F (about 49° C), ***and use a GLASS dish***. When warming etchant, I use a half-filled PYREX dish (to reduce the chances of spilling while carrying) and I place it into the microwave for a period of 2-3 minutes.

When etching PC board in sodium persulfate, it is not necessary to agitate the PC board during the etching process as the copper is dissolved into the solution. However, if using ferric chloride, it is STRONGLY suggested that you agitate the PC board as the copper which is etched away will precipitate out as cuprous chloride ($CuCl_4$ and Iron Oxide, FeO) which will lay on top of the bare copper, preventing the etchant from reaching it. Agitation will help to wash away the precipitate and to expose fresh copper to the etchant. I use a 6" (15 cm) long by 3/8" (1 cm) wide piece of scrap PC board with a notch cut into the end (see Figure 6) to catch and lift the edge of my PC boards and agitate them when I'm etching them. More recently, I will agitate the PC board in the etchant using a 10” (25.4 cm) long ‘PC board holder’, made from two 3/8” (9.5 mm) wide pieces of discarded PC board with small notches cut into the bottom end of each ‘stick’. See Figures 7 and 8 (left & right). This holder grasps the PC board more securely than does the single ‘agitator’ described above, and I can vary the amount of force exerted to hold onto the PC board by virtue of a ‘tension slider’ fabricated from a piece of metal cur from a heavy-duty paperclip which has been wrapped around the two 10” lengths of PC board stick.

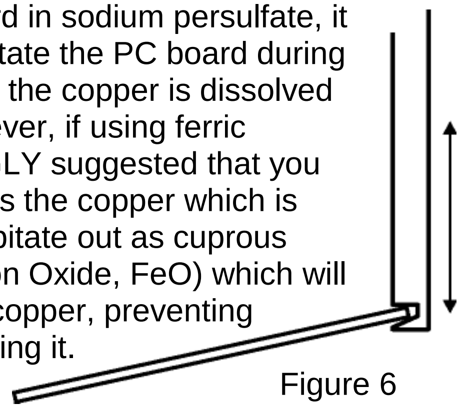


Figure 6

As the etching process proceeds, you can visually see the copper being removed. It is readily visible when using sodium persulfate, but more difficult to see with ferric chloride. You may want to lift the PC board out of the etchant and run a little clear water over it in order to more easily see how much copper remains to be etched away.

When the etching process is completed, remove the PC board from the etchant and flush it liberally with clear water to remove all traces of the etchant.

Figure 7

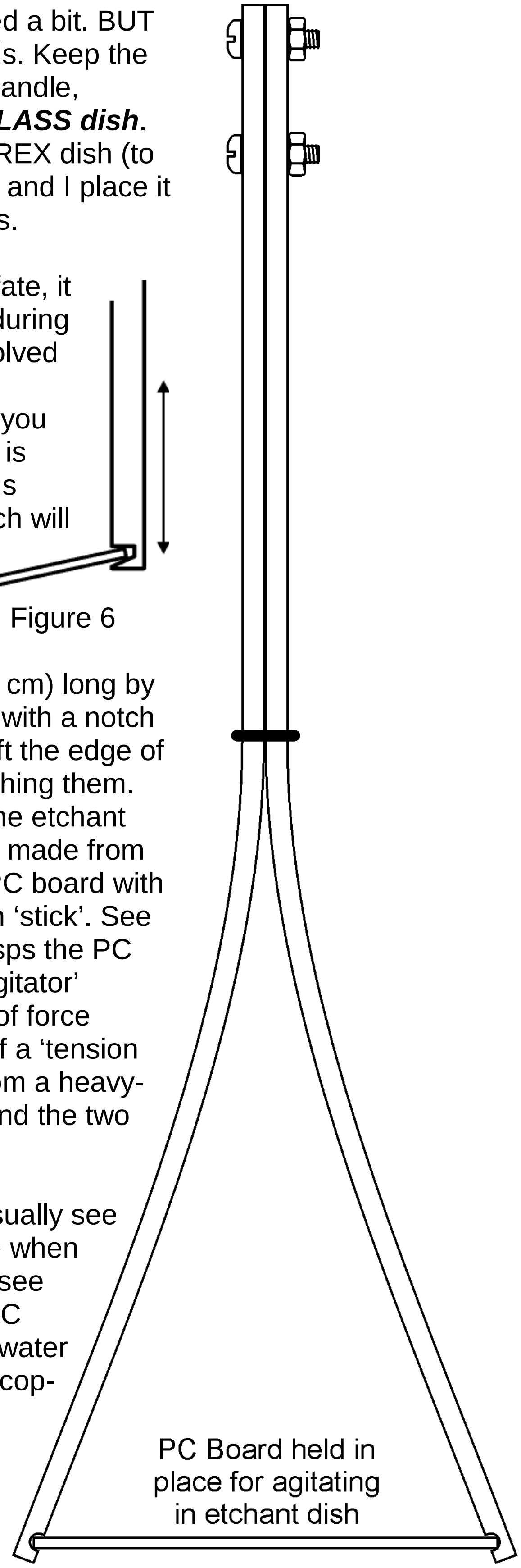


Figure 8

Return the etchant to a secure container (plastic or glass) and store in a safe place, away from children who might get into it.

At this point, the copper of the PC board is still 'protected' by the resist which must be removed before you attempt any soldering to the board. Actually, you *can* solder *though* the resist, but I don't recommend it. There are several ways to remove the resist. The easiest of which is the scour it off using kitchen abrasive cleanser and a cloth (or steel wool, a ScotchBrite pad, or some aluminum oxide sanding cloth). As you scrub, you will notice that the finish of the copper initially becomes a bit dull and then, as the resist is removed, it will become bright once again.

CUTTING & DRILLING PC BOARD

CUTTING PC BOARD

The substrate (material onto which the copper is bonded) of the PC board can be constructed of a number of various materials. The most common substrate is a fiberglass-epoxy mix (commonly referred to as G-10 Epoxy or FR-6 Glass-Polyester) which is very hard and can be quite difficult to cut or drill unless you have the right tools. G-10 Epoxy can be cut with a fine-toothed hacksaw blade, or a cutoff wheel in a DremelTool. PC board shears are also available, but these devices are usually very expensive to purchase. Most recently, some machine tool vendors, such as Harbor Freight, have offered multi-function machines which, though designed to work with sheet metal, will also work well when shearing PC boards. But, for the home builder, a hacksaw or a DremelTool usually works quite adequately. You WILL have to replace blades regularly however.

Other PC board substrates (e.g. epoxy composite (CEM-1), and bakelite) can be cut (and drilled) more easily than G-10 Epoxy.

DRILLING PC BOARD

Just as G-10/FR-6 PC board is difficult to cut, it is just as difficult to drill, especially with standard high-speed steel drill bits. Most **high-speed steel drill bits** will be dull after a dozen holes have been drilled, <u>*especially*</u> if the bit turns at a high speed. Use a fairly slow speed for the drill bit and you will have better luck, although the bit will still become very dull rather quickly.

Solid carbide drill bits are available on the used market (removed from PC board drill machines used by commercial PC board manufacturers, they will still be very sharp and quite usable), for 10%-20% of the cost of a comparable new bit, which might cost upward of $3-$4 (US). These bits, when run at HIGH SPEEDS (3,000-20,000 RPM) will drill hundreds (probably thousands) of holes before they dull. The problem with solid carbide bits is that they are terribly brittle and can break with only a small amount of lateral stress, such as catching the tip of the bit on the PC board as you move the board to a new hole.

I have had extremely good results using carbide bits in the range of #72 (0.025", 0.63 mm) to #60 (0.04", 1.02 mm). This range corresponds to wire gauges between #22 and #18. Of course, other diameter drill bits would work well, too. These just happen to represent the range of wire sizes I most frequently use... skewed toward the smaller end of the range (0.025" through 0.035").

The drill (and bit) MUST be held steady while drilling or you *will* break the bit, guaranteed. I have found that a DremelTool, in the Dremel Drill Stand works very well for drilling PC boards. Sears also offers a somewhat larger drill stand which is specifically designed to accommodate both Sears and Dremel hand-held motor tools. This stand also offers a much deeper 'throat' than the

Dremel stand, so you can drill larger PC boards when using the Sears stand. While this stand has a small amount of lateral movement, it is generally stable enough to allow you to drill without damaging solid carbide bits. Although I have not seen it, I understand that Dremel now offers a new (supposedly more stable) drill stand (Model 220 WorkStation) for their newer hand-held DremelTools. I am not sure it will accommodate older models of the DremelTool however.

Once I have etched, cleaned, cut, drilled, and subjected my PC board to other unmentionable horrors, it is now ready for building that special circuit for which it was created. I'm now ready to sling solder!

RECENT DEVELOPMENTS…

I have recently become aware of **FREE** schematic and PC board layout software available from ExpressPCB (http://expresspcb.com). Although this software is specifically designed and written to require you to submit your designs to ExpressPCB, for them to make your PC boards, I have found that I can grad a screen-capture of my layout (using Windows' Screen Print feature), PASTE this image into a graphics program, re-size (resample) the image to the size I want, then import it into CorelDraw, and overlay my CorelDraw objects onto the layout created in ExpressPCB. This allows me to more rapidly create a new layout (using the ExpressPCB software) and then to create the final design using CorelDraw. If you are interested in more details of exactly how I accomplish this, please feel free to ask. Unfortunately, the Express PCB software will not allow you to send an image of your PC board design to a printer (or to a bitmapped graphics file). But it will allow you to print the schematic and the PC board parts layout. If you want a bitmapped graphics file of your layout, you must use some form of screen capture software. The process is still faster than creating the design (at least if it is the least bit involved) using CorelDraw alone.

Some information regarding etchants follows on the subsequent pages.

Ferric Chloride ($FeCl_3$)

Toxicology

Toxic by ingestion, strong irritant to skin and tissue.

Protective Equipment

1. Ventilation - Work area should be properly ventilated.

2. Clothing - Appropriate clothing to prevent contact of the liquid with the skin; Use Rubber gloves to protect hands.

3. Eye - Protection - Use safety glass or goggles to prevent eye contact with the solution. An eye wash should be available in the work place.

Good Points:

Generally lower cost than sodium persulfate.

Usually lasts longer than Sodium Persulfate... etches more copper before it must be discarded.

No significant shelf-life. Lasts almost indefinitely, until depleted by etching.

Bad Points:

Oxidizer

Reacts ***VIOLENTLY*** when it contacts aluminum!

STAINS skin and clothing! The good thing is that the stain to clothing doesn't last long since the stained cloth will usually fall out, leaving a nice *hole* where the stain once was. Fortunately, stained skin doesn't leave holes... although skin *can* become irritated by the presence of this chemical.

Difficult to dispose of properly. You will usually have to find a local State-approved hazardous waste disposal facility to accept spent $FeCl_3$.

> I have been told that $FeCl_3$ can be mixed with Quick-Crete and then buried once it has hardened. But I cannot confirm this as an acceptable substitute for proper disposal.

Sodium Persulfate ($Na_2O_8S_2$)

Toxicology

Harmful if swallowed. Very destructive of mucous membranes. May cause dermatitis. May cause irritation. May cause sensitization

Protective Equipment

1. Ventilation - Work area should be properly ventilated.

2. Clothing - Appropriate clothing to prevent contact of the liquid with the skin; Use Rubber gloves to protect hands.

3. Eye - Protection - Use safety glass or goggles to prevent eye contact with the solution. An eye wash should be available in the work place.

Good Points:

Doesn't seem to stain

Available as dry crystals which can be mixed with water when needed.
When dry, shelf life appears to be relatively unlimited.

Bad Points:

Oxidizer

Limited shelf life when in solution - usually several months, max.