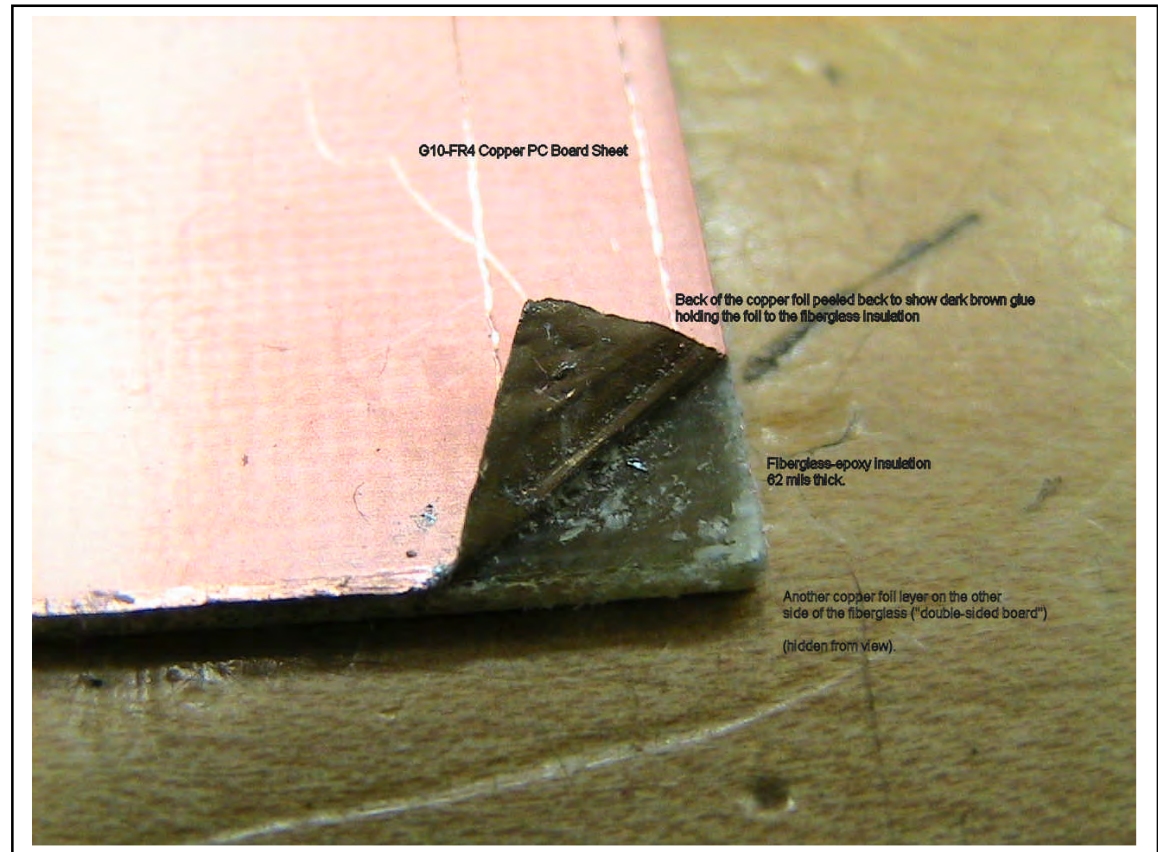


VII PCB Fabrication

A. PCB Material

1. Very early in the electronic business, we tried literally to make printed circuit boards. That is, we took an insulator (bakelite was all we had at the time) and tried to print copper wires ("traces") onto the bakelite plastic using metal-loaded ink. The problem was that the ink wasn't rigid enough to survive the slightest bit of flexing of the bakelite, nor did it take well to vibration.
2. The next step in this process was to take that same sheet of bakelite and glue a very thin metal foil to the plastic surface. The desired circuit was drawn on a sheet of paper and a photographic silk screen process made from that paper artwork screened acid-resistant paint onto the metal (almost always copper) foil in the pattern drawn on the paper. The whole plastic-foil-paint combination was then dropped into an acid bath and the unwanted copper was dissolved in the acid. Although technically an "etched board" the original "printed circuit" board name was retained.
3. Over the years refinements have been made to the process but the basic material remains the same -- an insulator covered with copper foil on one side of the insulator (single sided board) or on both sides of the insulator (double sided board).
4. While many insulators have been used over the years for pc board material (phenolic, paper, rubber, Teflon®, Mylar® and other exotic materials) for most boards made today the insulator of choice is fiberglass-epoxy resin. We call this material G-10 or FR-4 interchangeably, although FR-4 is the Fire Retardant grade of G-10 if you

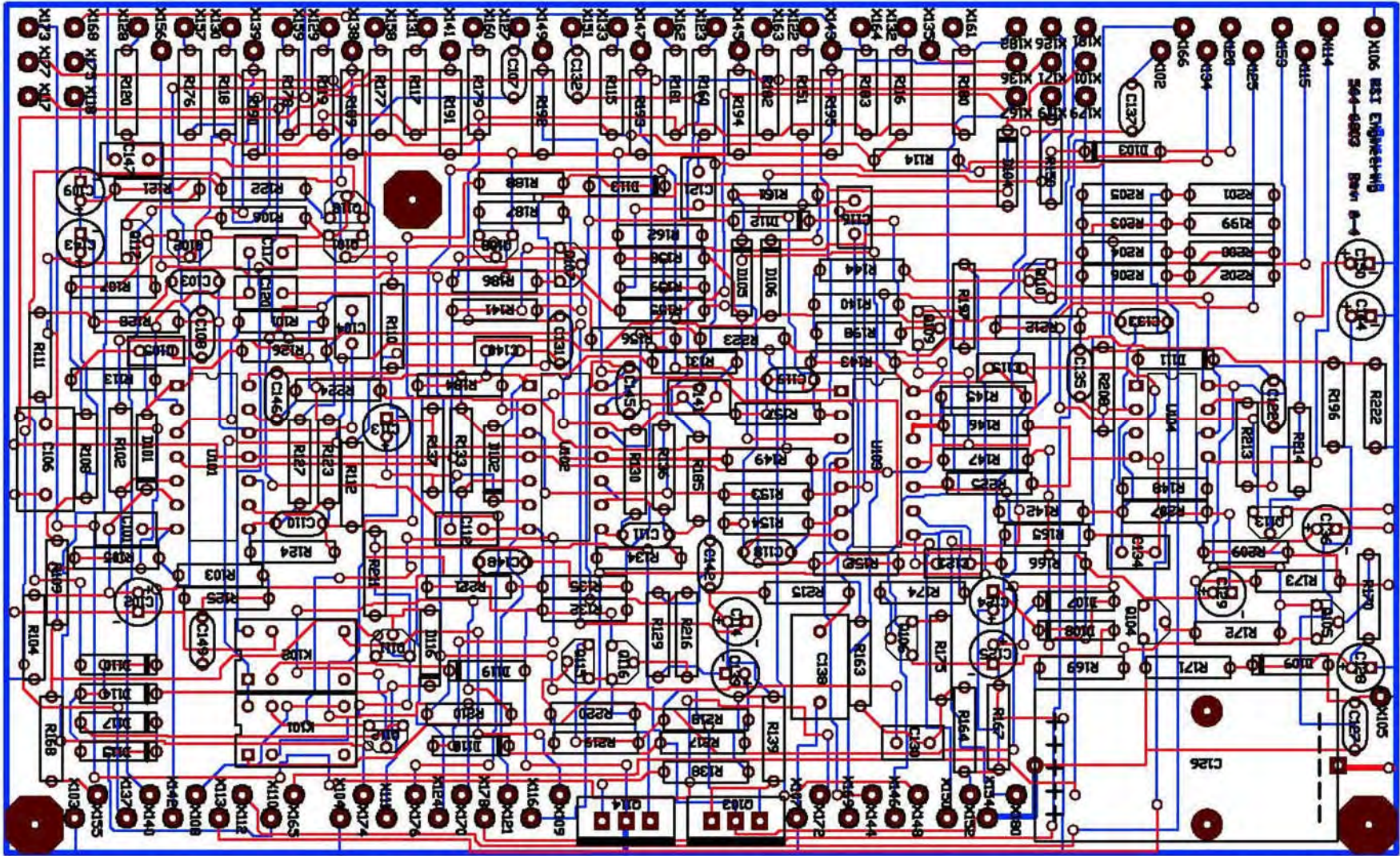


want to be technically correct. G-10 comes in many thicknesses, but 62 mils is the "standard" pcb material.

5. Similarly, the metal foil can be any metal, although copper is the universal metal of choice. It comes in many thicknesses but "1 ounce" copper 14 mils thick is the standard.

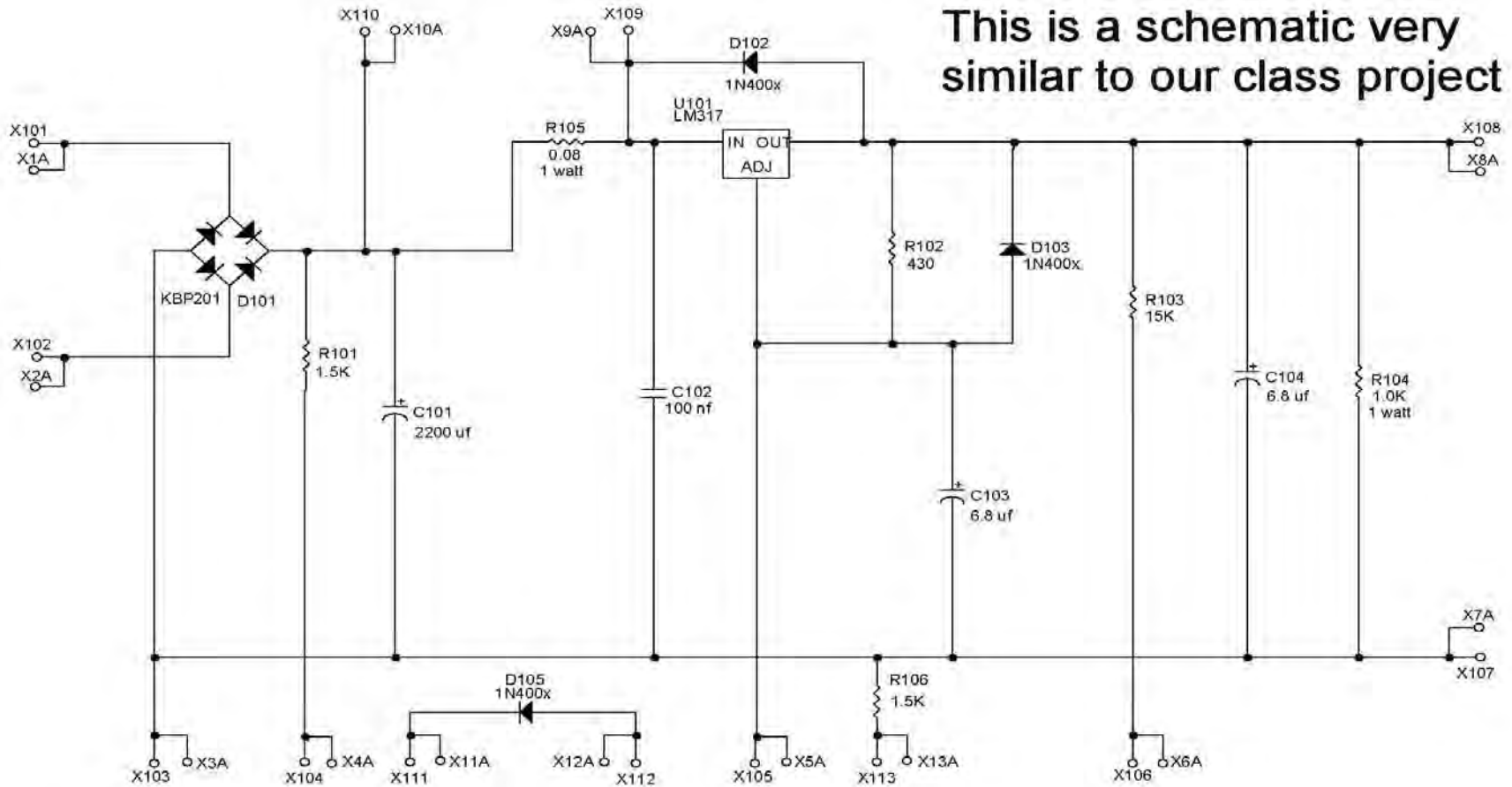
B. Layout

1. As we noted above, the original "layout" artwork was done using a silkscreen process derived from a paper ink drawing of the desired circuit. A little later on down the development trail we began using clear plastic instead of paper and black tape along with preprinted black tape patterns for the traces and component hole "pads". This latter process was called "tape and donuts" for the shape of the most common pad pattern -- a small black round shape with a clear hole in the middle for a drill bit guide.
2. Along about the same time that tape and donuts came along, also came along a method whereby a photographic acid-resistant emulsion could be laid down on the copper foil, the pattern generated by the artwork could be photographed onto a film negative, and the resultant pattern could be photographically transferred to the emulsion. The emulsion could be developed, the undesired exposed emulsion washed off, and the desired pattern then chemically hardened and tossed into the acid bath to etch off the unwanted copper.
3. When computers came along, we started getting rather sophisticated with how we make pc boards today. It is now common practice to use the same program to draw the circuit (called "schematic capture") and transfer that schematic design directly into a pcb drafting planning tool called a "pcb layout" routine.
4. A further refinement of the pcb layout routine called "autoroute" will actually lay the components out on the board and automatically route the wire traces between the pads as shown on the schematic. Finally, files called "gerbers" (named for the company that invented them) and numerically controlled drill files (called NC files) are made from the pcb artwork. These files are electronically sent out for manufacture to a company with cameras and automatic drilling equipment that take those gerbers and NC drill files and automatically flash the layout onto the emulsion, drill the holes, etch the board, and it all comes out the other end of the machine never having been touched by human hands.
5. While this process is all well and good for production boards, the home hobbyist, the R&D engineer, or the student who only needs one or two boards for prototype use can still use the computer to do the schematic and pcb layout, but shortstop the process before it gets to the gerber stage and print out the circuit on a plain old laser printer for making boards themselves (see C.4. below).



This is an example of a commercial product circuit board laid out in Traxmaker. The blue lines are bottom traces, the red lines are top traces, and the black outlines are the component reference designators on the top overlay ("silkscreen").

This is a schematic very similar to our class project

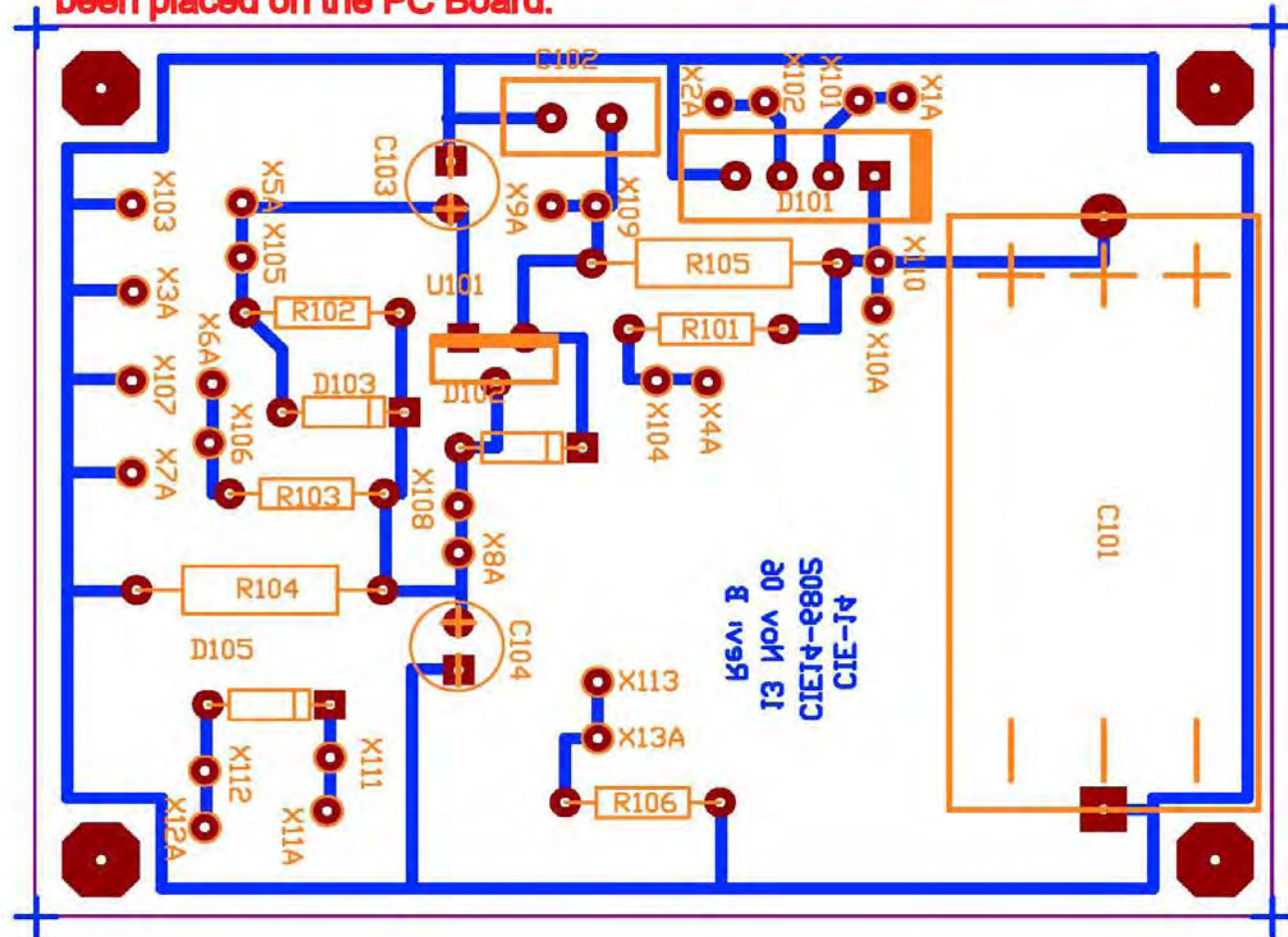


Notes:

1. All components will be found in the parts bin labeled "Electronic Fabrication". Do NOT change the labeled value of any of the parts.
2. R105 is wound with 10" of #28 wire on a 1K 1 watt resistor form.

CIE-14 Power Supply (Basic Version)	
PC Board Components to make PCB CIE14-6802	
Rev	ID
C	CIE14-5802
Date: 27 Nov 06	Page: 1 of 1

This is the PC Board made by exporting the Circuitmaker schematic to the Traxmaker layout program. Note that every component on the schematic has been placed on the PC Board.



C. Resist

1. Most any organic substance will work quite well for an acid-resistant etch -- paint, photographic emulsion, ink from a "permanent marker", and even good old oil from your hands will work quite well.
2. One of the organic substances that has been used quite successfully in prototype manufacture of pc boards is laser "ink". This ink is actually just a plastic that has been loaded with black dye. When a laser printer "prints", it actually just lays a pattern of plastic down onto the paper and then melts the plastic into the pores of the paper. (Did you ever wonder why the paper comes out of the laser printer warm to the touch?)

The process for prototype resist is to print the desired pattern onto a slick paper that does not let the "ink" soak into the paper, then transfer this pattern onto the surface of the foil. Since the plastic ink is impervious to acid, the pc board can then be directly etched with an acid etch. This "paper" is actually a photo gloss paper intended for inkjet printers and with a little work, we can

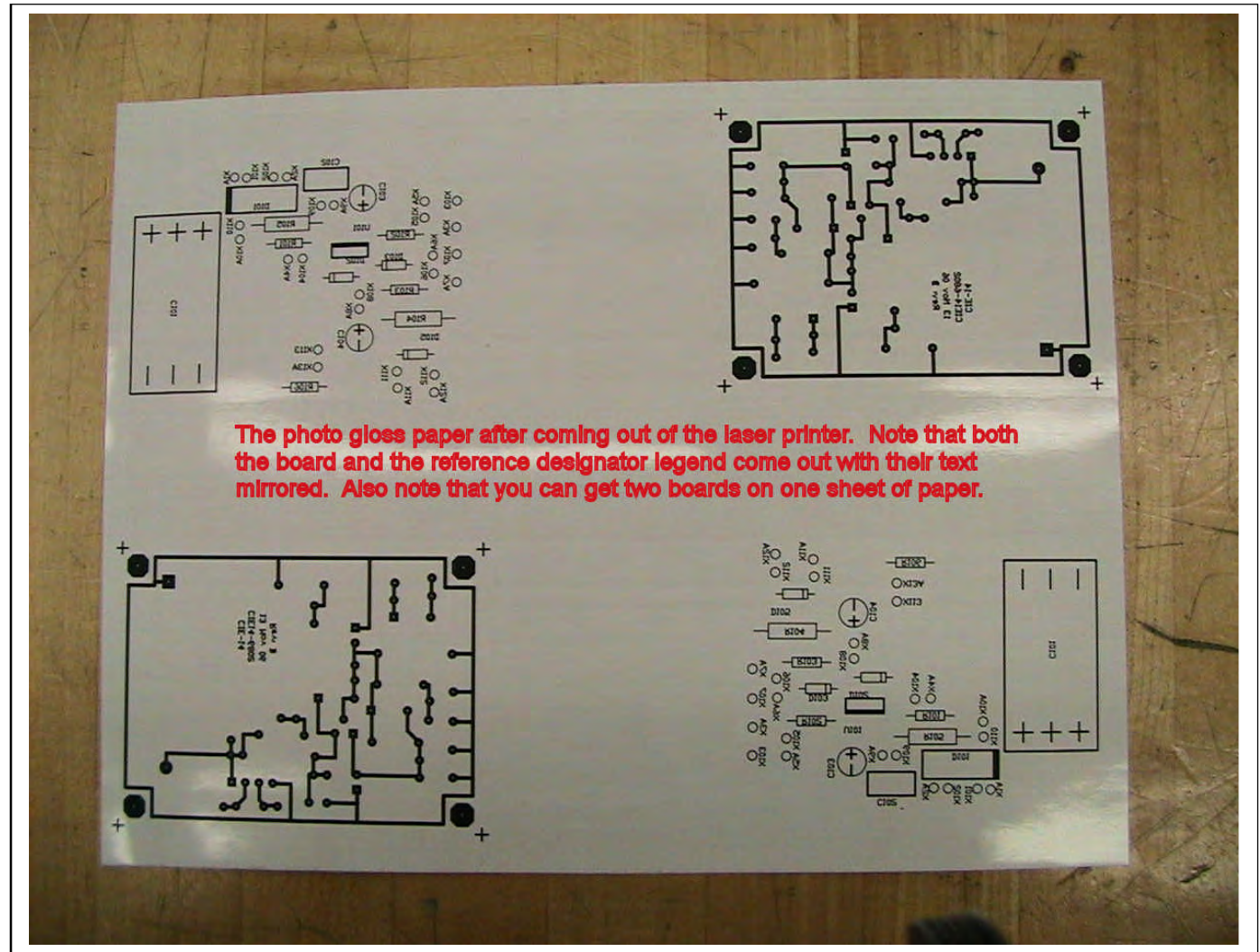


transfer this pattern directly from the paper to the copper surface by remelting the ink from the paper to the copper foil.

3. Here are the steps for making a "laser printer" pc board:

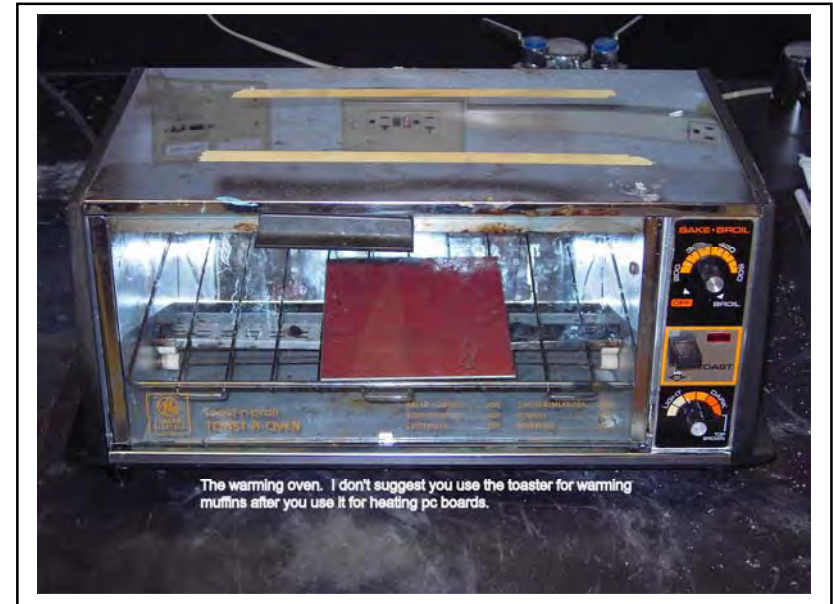
- a. Clean and microscratch the pc board foil using an abrasive powder copper cleaner ("Copper-Brite®" available at the supermarket) and a heavy duty nylon pot scrubber "green pad". Wash the board thoroughly in warm water. Do not touch the surface of the board from this point until it goes through the acid bath.

- b. Laser print the desired pattern onto photo gloss inkjet paper. Be sure that the pattern has some mirrored words on it so that you can see instantly "Mirrored" means that the pattern is printed on the paper as though it was viewed in a mirror. When you turn the paper over to put the pattern on the board, this mirroring will be reversed and a non-mirrored pattern will be placed onto the board.

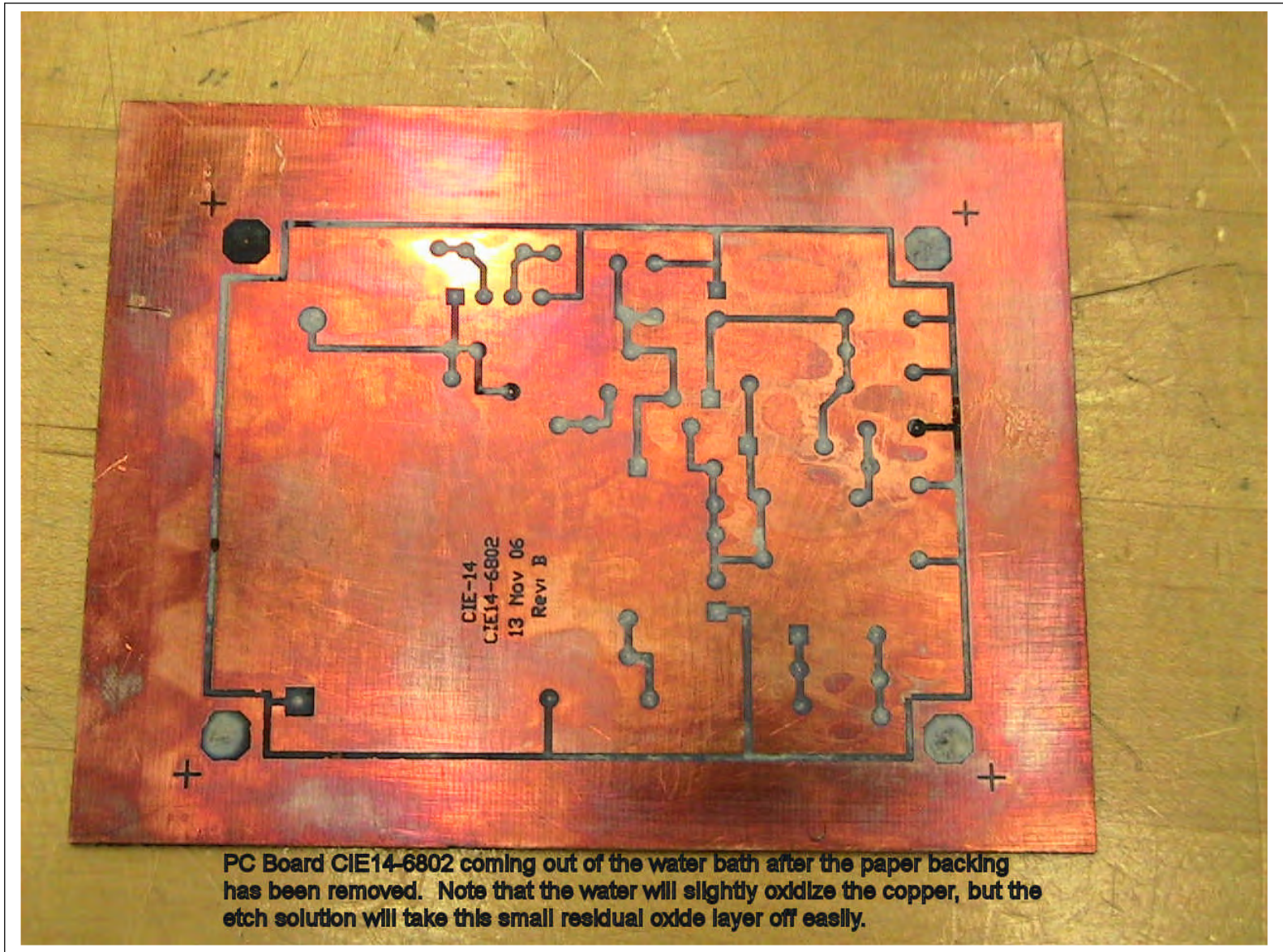


Dry the board thoroughly and place it in a warm oven at about 120 degrees for no more than two minutes (a toaster-oven works well). The board must be perfectly dry for the toner to transfer.

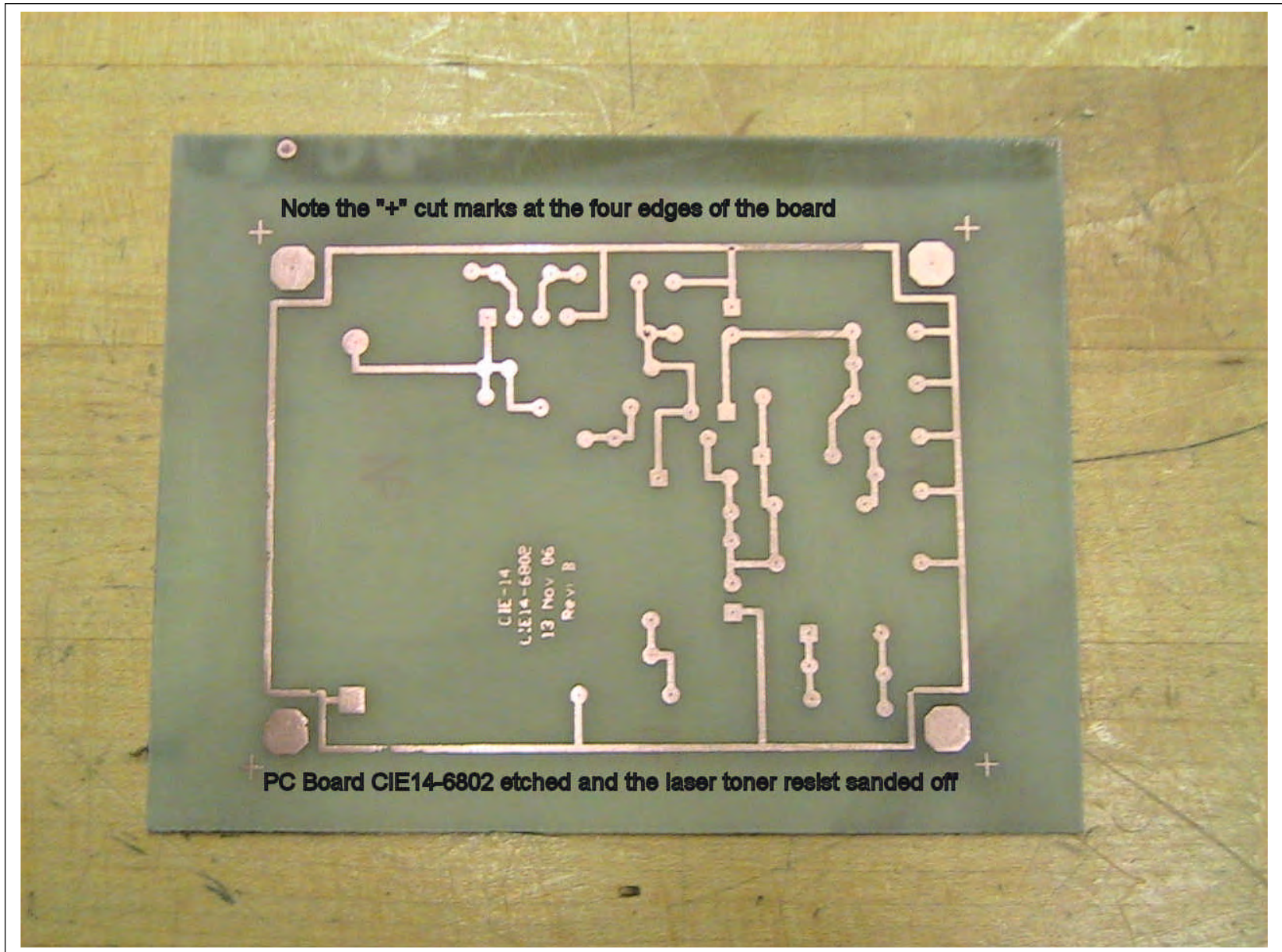
- c. Take the board from the oven and place the laser printed pattern directly onto the surface of the board. Do not allow the board to cool before placing the pattern onto the board.
- d. Immediately take a hot clothes iron set at the highest temperature and begin "ironing" the pattern onto the board using heavy pressure and move the iron like you are ironing a shirt -- back and forth, side to side. After two or three minutes (depending on your particular iron) you should start to see the resist pattern showing through the back side of the photo paper. When the pattern is clearly discernable (no more than 5 minutes) the pattern has been "ironed" onto the pc board.
- e. **CAUTION** the board is very hot. Transfer the board to a warm (**NOT HOT**) water bath. In about 15 to 30 minutes you will see the paper backing begin to lift from the surface of the board. When the bubbles of paper are almost all across the surface of the board, it is time to **GENTLY** begin to peel the paper backing away from the board (do the peeling under water). This is generally done in two or three stages. First the heavy paper will peel off of the back,



then the lighter front coating will peel from the board, then the small bits that didn't peel with the other two steps can be peeled from the surface of the board. Don't worry about any of the paper still stuck to the laser ink on the foil surface. You now have a pc board ready for the etch bath.



- f. After the board is etched, the resist can be removed from the copper by means of 80-120 grit sandpaper (for hand sanding, wet-or-dry generally works best under a warm to hot water bath) . A small electric pad sander will make the job much easier.

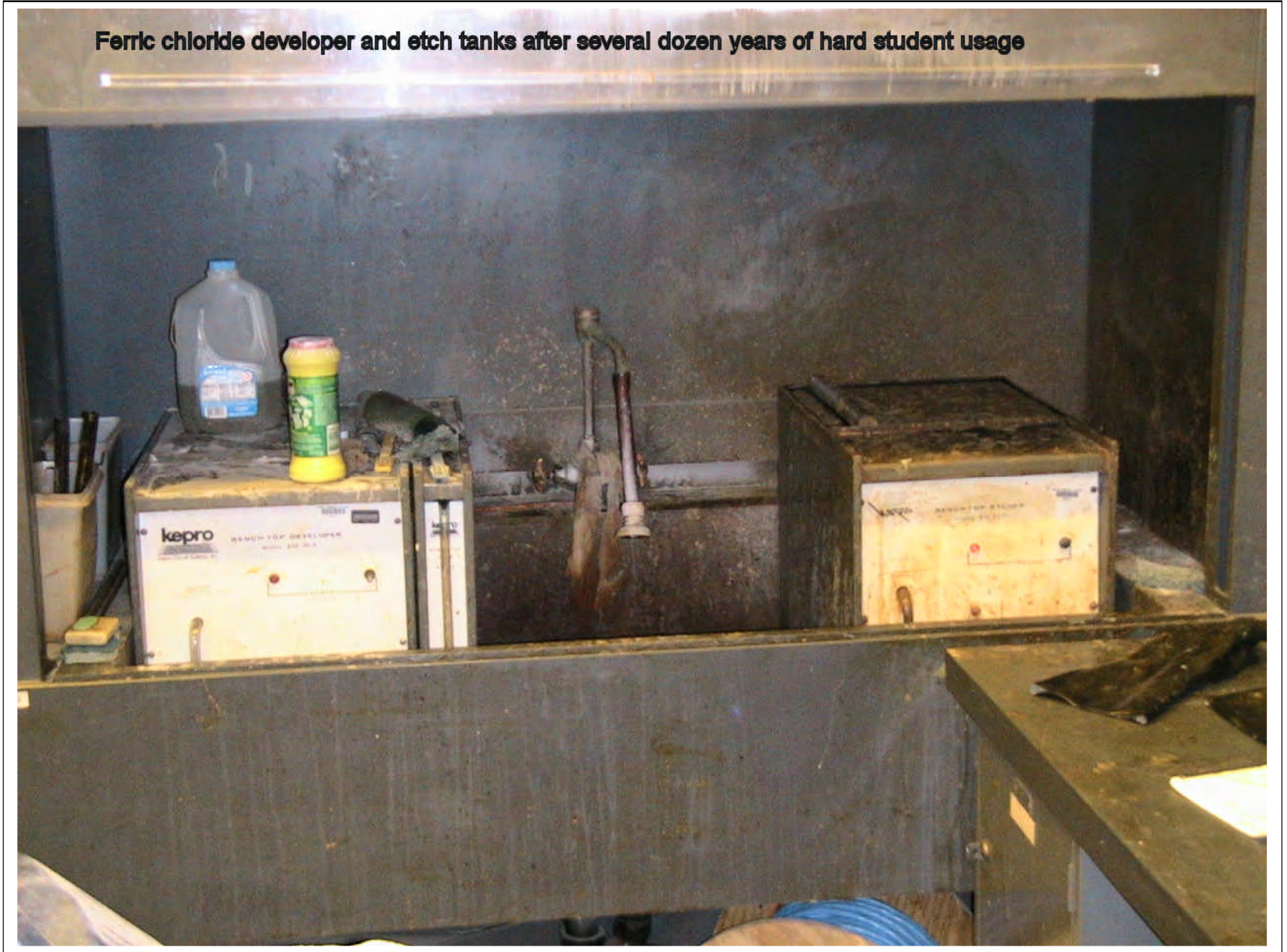


- g. It is possible to make double-sided PC boards this way.
- i. Before etching, drill two SMALL holes through the bottom resist, the bottom copper, and the top copper surfaces. These two holes should be at diagonal corners of the board to the extent possible in order to make registration easier (you may wish to place special "registration holes" at diagonal corners of the board strictly for this purpose). Make sure that there are overlapping holes on both the bottom and the top resist patterns and that they are exactly aligned.
 - ii. Print out the top resist pattern on photo gloss paper MIRRORED. Make a small pinhole through both of the paper registration holes as close to the center of the hole as possible. Poke a small wire up through the bottom resist and place these pinholes down on the top of the board so that the paper makes good contact with the top copper surface of the board.
 - iii. Thermal management is critical. The idea is to get the top copper/paper as hot as possible without melting the bottom traces. One technique that has proven itself useful in this management is to freeze a piece of aluminum larger than the PC board in the ice chamber of a refrigerator. Lay the bottom side of the pc board on this frozen aluminum while ironing & towel-pressing the pattern onto the top copper side. Use the minimum heat necessary to transfer the pattern to the top side.

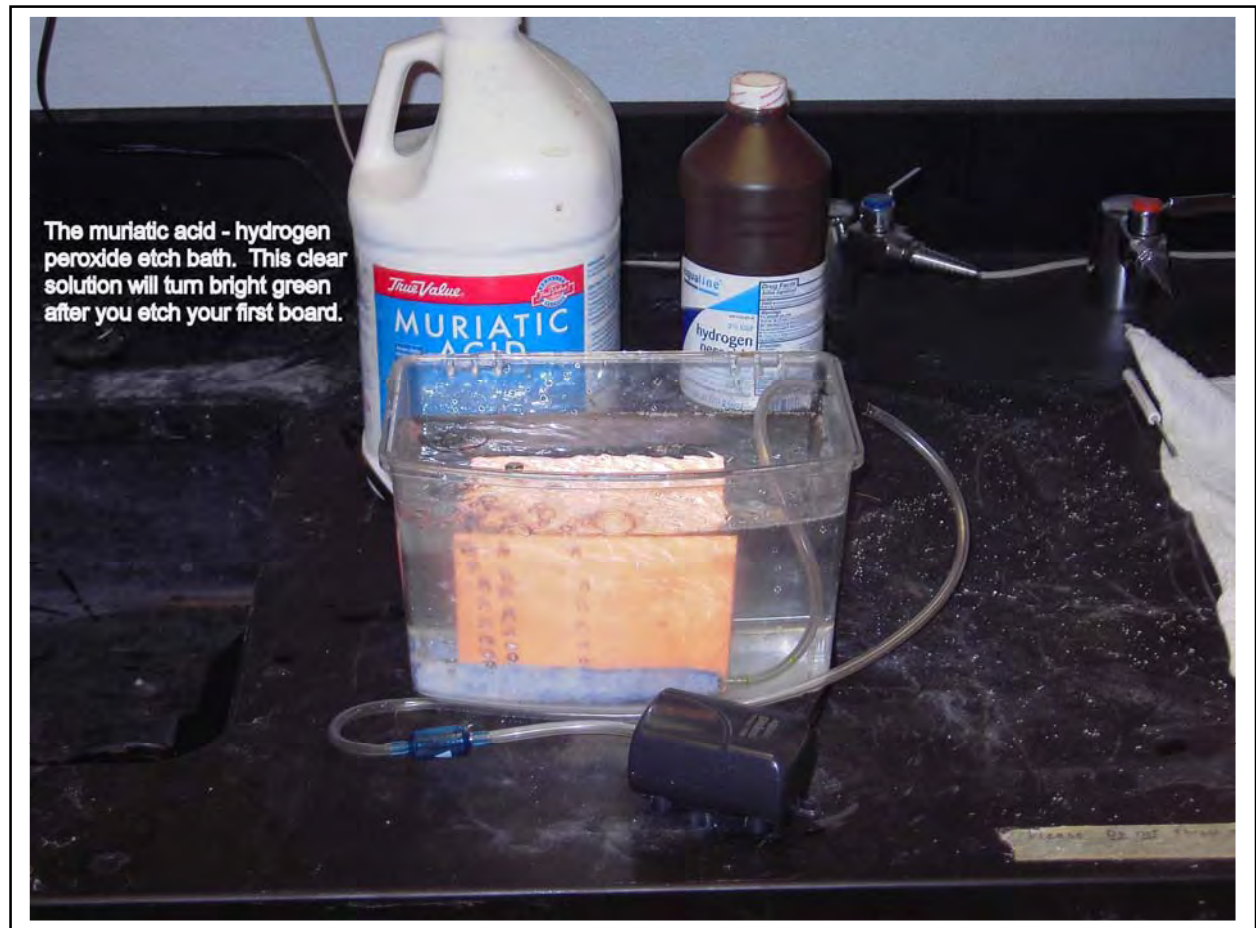
D. Etching

1. Over the years, we've used many substances to remove copper from the pc board material. Three compounds have evolved that have shown themselves to be excellent at this job. They are, in the order most common in industry, ferric chloride (FeCl_3), ammonium persulfate ($(\text{NH}_4)_2\text{S}_2\text{O}_8$), and cupric chloride (CuCl_2).
2. Ferric Chloride is a brown acidic liquid that is obtainable from many of the large electronic supply houses. It readily stains anything organic that comes in contact with it (clothing, skin, etc.) a permanent rather ugly brown. It is considered a hazardous material and commands a premium shipping fee. The good thing about ferric chloride is that it is very fast acting as an etchant and made even faster by the addition of a small amount of heat (no more than 120°F). There is no known "home remedy" to remove the copper from the ferric chloride, so once it is "loaded" with copper it must be disposed of. This is not as simple as it sounds, as loaded ferric chloride is a very toxic hazmat. Some say that you can neutralize the acid with an alkali such as Drano or lye and then use the resulting solution to make a "concrete cage" with common cement. This merely postpones the problem for a few generations until the concrete begins to deteriorate and leaches the iron and copper back into the groundwater.

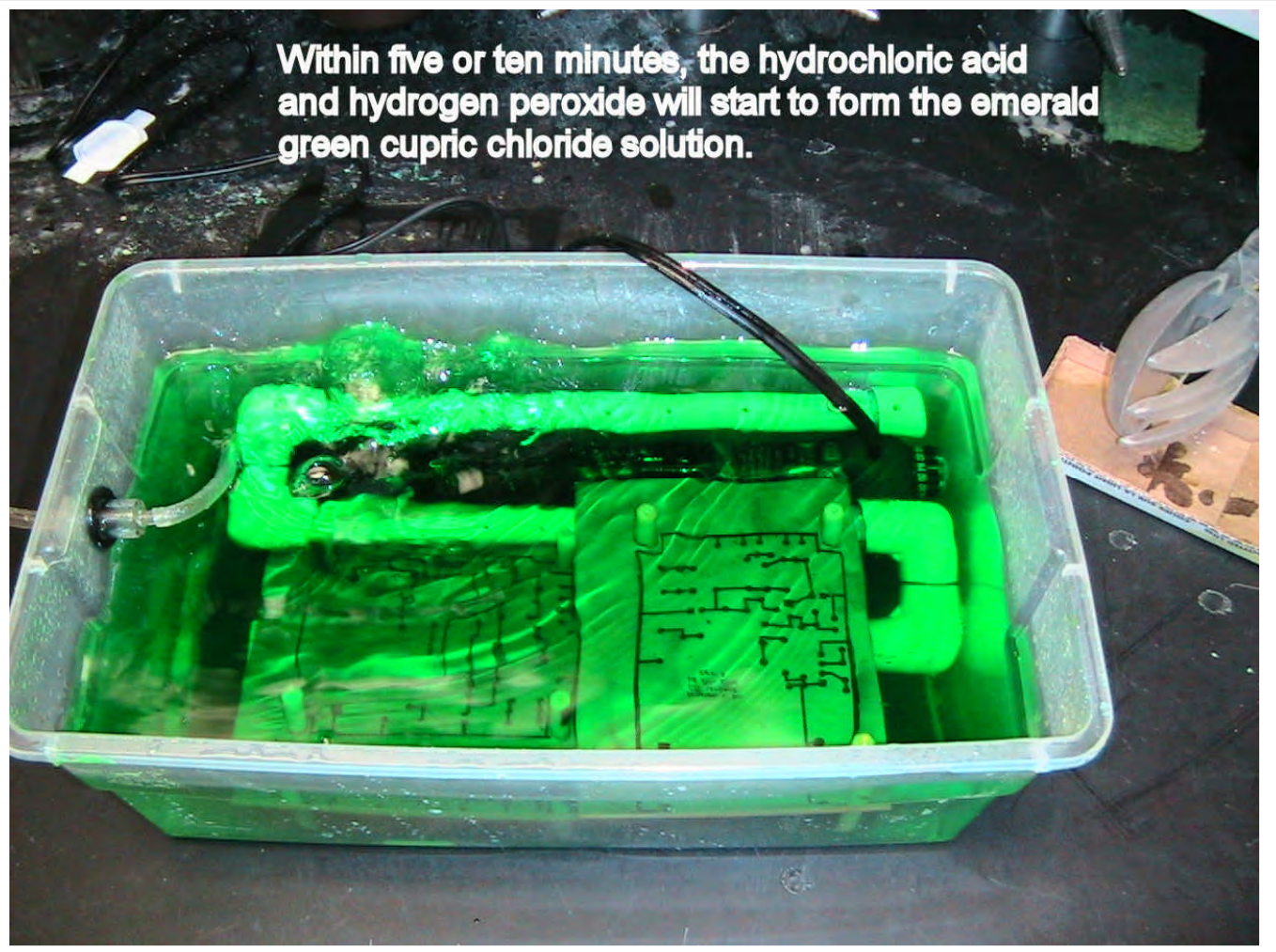
Ferric chloride developer and etch tanks after several dozen years of hard student usage



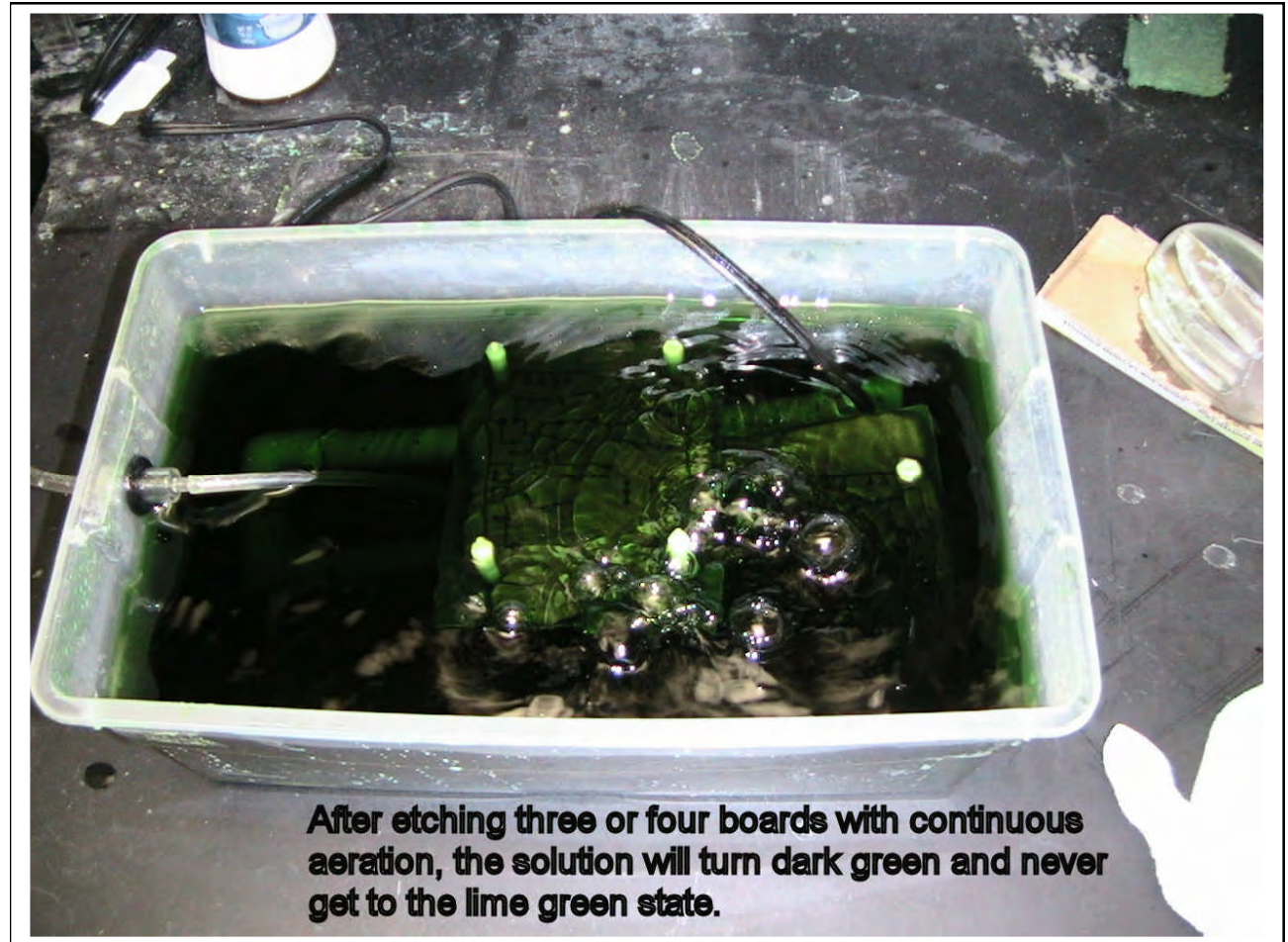
3. Ammonium persulfate is a white alkali crystal that is mixed with water to form an alkali etch bath. It used to be available from the large electronic supply houses, but has fallen from popularity for some reason. When mixed it does have a shelf life of a few weeks at most. Most of the comments regarding ferric chloride (with the exception of the brown stain) are equally valid for ammonium persulphate.
4. Copper chloride is a good candidate for prototype etching in that it can be obtained at the local home store and is fairly easy to use and dispose of when done. However, the process should be carried out either under a fume hood or outdoors.
 - a. The basic chemicals used in the process are muriatic acid (28% hydrochloric acid rock cleaner from the home improvement store) and hydrogen peroxide from the drug store. (The hydrogen peroxide is just used once to get the process going or to speed the regeneration process.)
 - b. Mix the muriatic acid and hydrogen peroxide together in a glass or plastic container like a fish aquarium. Use four parts of water to three parts of muriatic acid to one part hydrogen peroxide.
 - c. Use a plastic "fish bubbler" and aquarium pump to keep air moving through the etchant during the etching process.



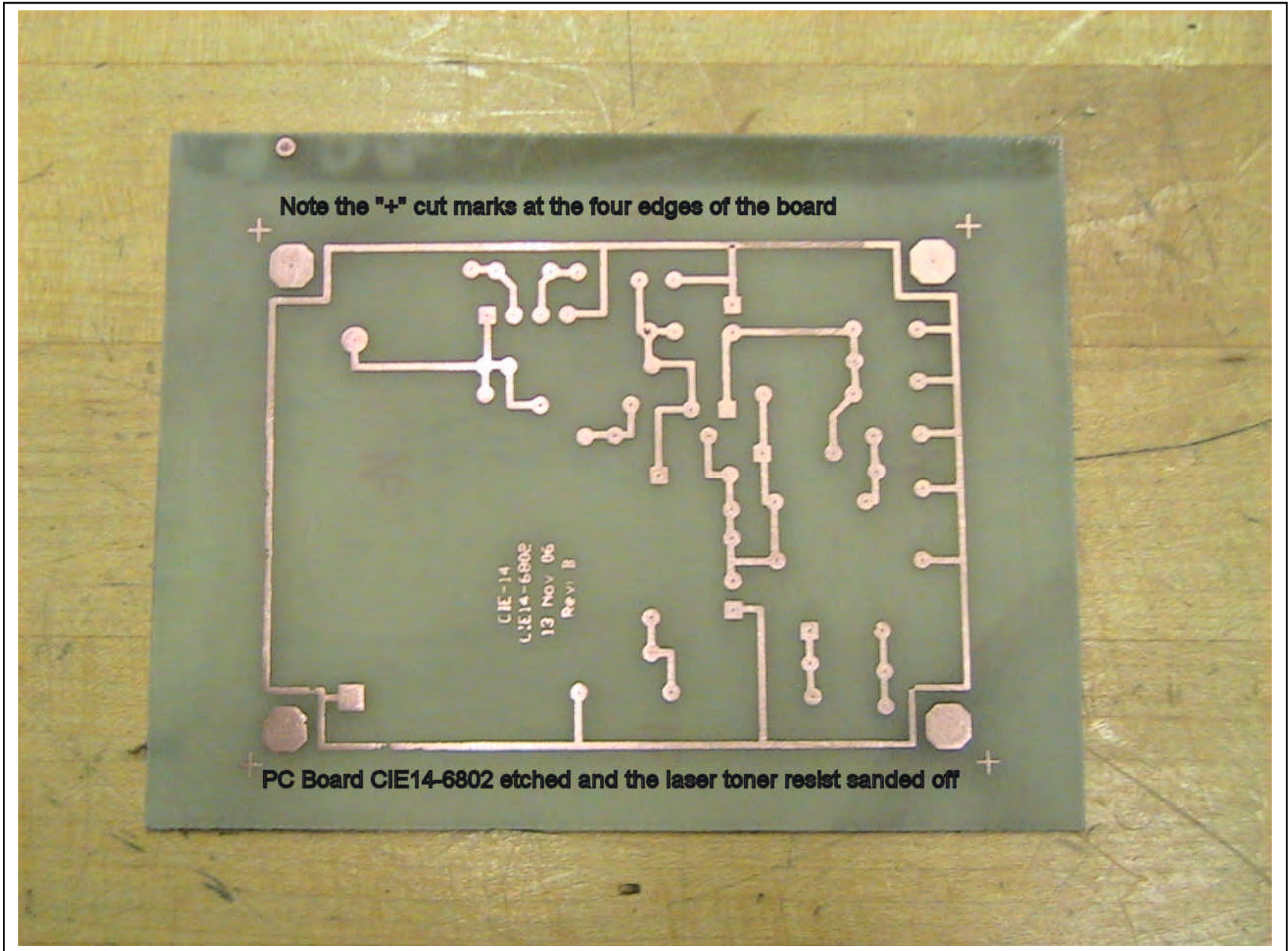
- d. Put the board to be etched into the etch bath. In a short time you should see the copper turning from a gold color to a dull red. In 20 minutes or thereabouts all of the copper not protected by the resist should be eaten away and the etch solution should be either a light emerald green or a bright lime green (or a dark green if you are using continuous aeration). Bring the solution back up to the same level as you started with by the addition of water and acid in a 1:1 proportion or removal of excess etchant (see g. below).
- e. The muddy dark green is cuprous chloride which is formed by the action of the hydrochloric acid on the copper. However, continually bubbling air through this cuprous chloride will oxidize it to the emerald green cupric chloride, which can be reused to etch another board, form the cuprous chloride, be reoxidized back to cupric chloride and so on indefinitely. When etching a square 6" x 6" board, the cuprous chloride vigorously agitated by a fish bubbler will be oxidized back to cupric chloride in a few hours.



- f. If you are not using a bubbler during the etch process, and especially if you are doing multiple boards in one session, the color will never get from the emerald green to the lime green stage. Instead, the solution will continue to darken until it is nearly opaque and the etching process will slow greatly. At this point, the addition of hydrogen peroxide as an oxidizer will return the solution to the bright emerald green state and the etching will continue at a normal rate.

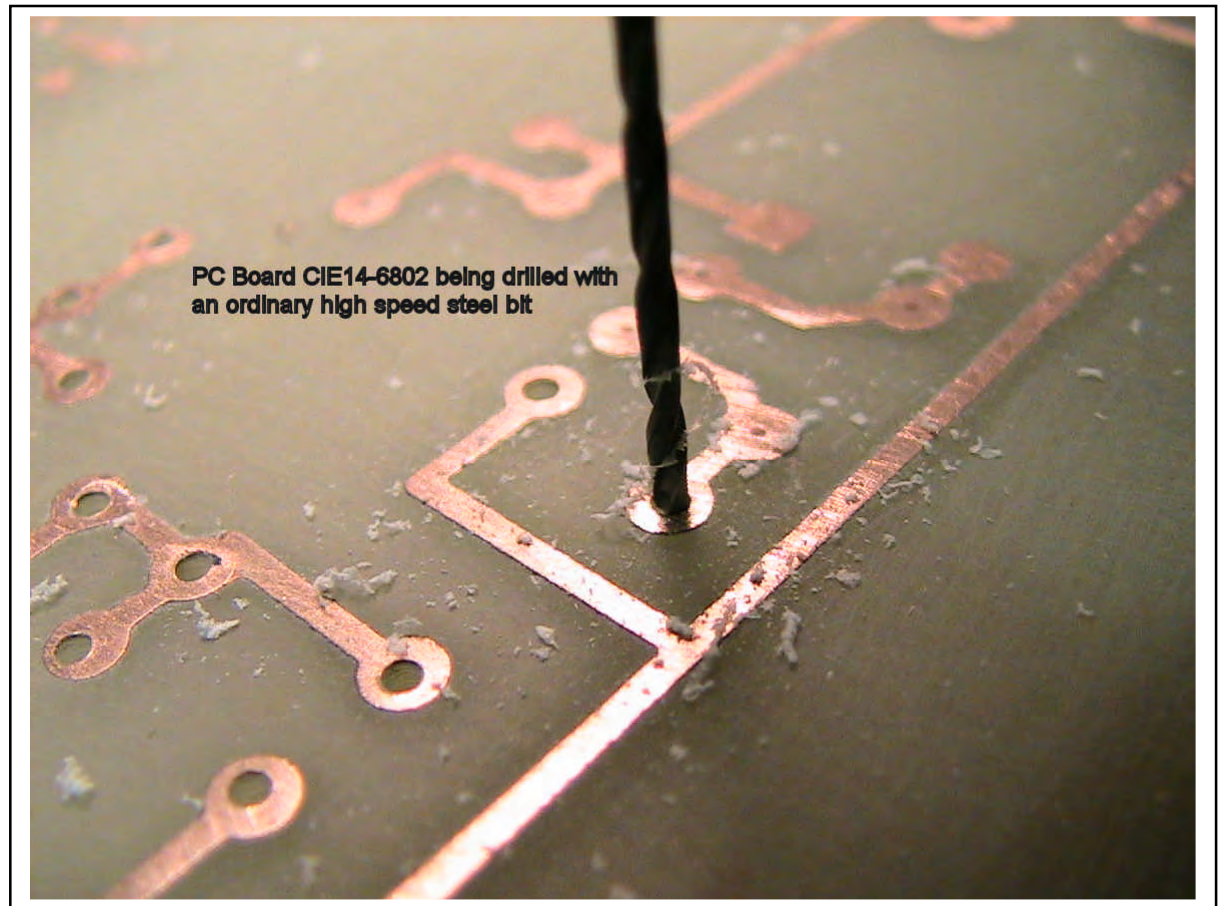


- g. The problem is that after a few dozen etches, you will have more solution (due to the water and acid addition) than you started out with. However, the solution may be easily disposed of by simple neutralization and metal plating. While a discussion of neutralization is beyond the scope of this text, you will find that common drain cleaner ("Liquid Drano®") will work well to neutralize the chloride acid and common swimming pool testers ("Phenol Red") will indicate a neutral solution. Then a steel wool pad immersed for a day or so in the neutralized solution will cause the copper to plate out on the steel wool, which may then be disposed of as scrap metal. The now neutralized and deionized liquid solution (mostly common salt water) is relatively benign.
- h. Or, you can simply store the removed cupric chloride in plastic or glass jugs and donate it to somebody else who wants to start making their own pc boards.



E. Drilling

1. In general, you can get by with three or four sizes of hole. Small parts (quarter-watt resistors, transistors, through-hole ICs, up to #24 wires, etc.) can be done with an 0.035-0.041" drill. Larger components and bigger wire can be done with an 0.045-0.051" drill, and mounting holes (#2 or #4) can be done with an 0.120" drill. Sacrifice a relatively loose ("sloppy") fit for a minimum number of bit sizes -- the solder will easily fill up a 10 mil annular gap.
2. You can use plain old high speed steel ("hss") bits if you wish, but the fiberglass will make short work of them. You might get a hundred holes with a single hss bit. A carbide bit is far more expensive, but will last a thousand holes or so.



One tradeoff you may wish to

consider is that a relatively inexperienced person, especially with a manually operated drill press, will tend to snap the more brittle carbide bits more often than the softer yet more pliable hss bits. Experience has shown the cost to be roughly equal between a student using either hss or carbide bits. You can tell when a hss bit is nearing end of life by examining the copper around the hole -- a dull bit will tend to give a burr much like a hole in aluminum will give a burr with a dull bit. (The technique for removing pcb burrs is identical with aluminum -- a larger bit will remove the burr quite nicely.)

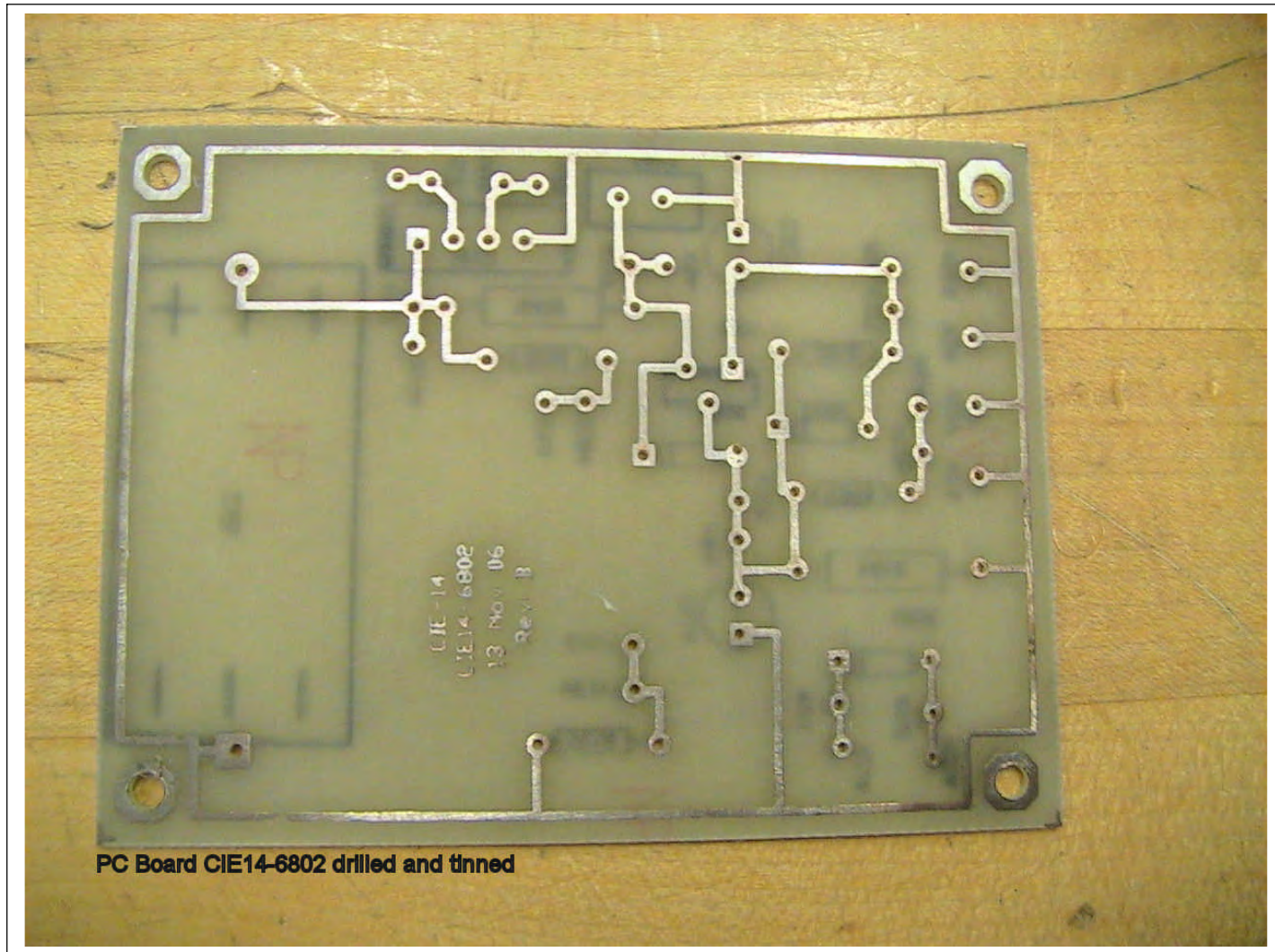
3. You can use a normal "metal drilling" drill press on its highest speed, but for better hole drilling in pc board material (less burrs, better centering, etc.) a high speed drill press meant specifically for pcb drilling is recommended.

F. Plating

1. PC board copper, left to itself, will oxidize and sulphate when left exposed to air. While this oxidation and sulphation does not materially affect the conduction of electricity of the underlying copper, it does make soldering a challenge. It is also true that bare copper needs to be "tinned" by the action of the solder and flux when making a joint directly onto copper pads.
2. Several companies make tin plating solutions that plate by simple immersion at room temperature (Google "Tinnit"). The tin plating thus formed is very thin (0.1 mil or so) but it will keep the copper from oxidizing plus give a very good surface for soldering.

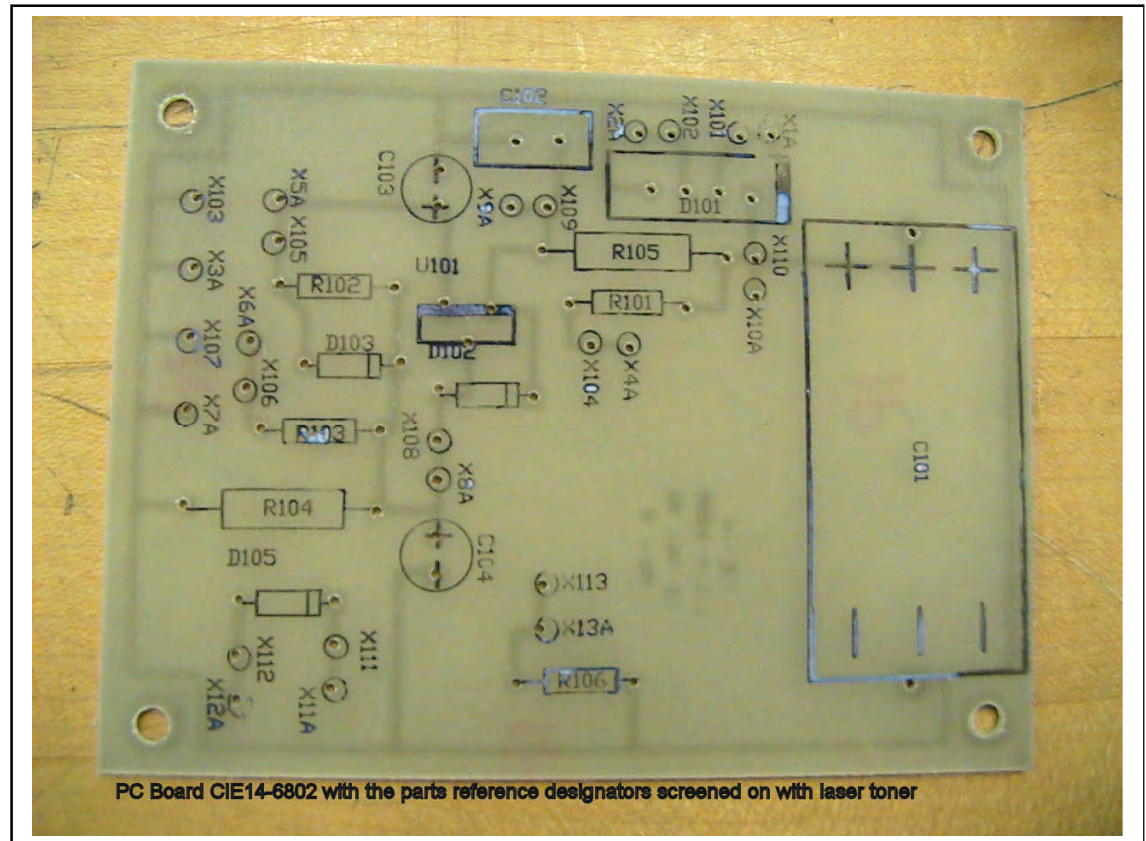


3. One problem with simple immersion plating is that the tin, too, will form an oxide after a while. While flux will strip tin oxide, it is best that the board does not sit around unsoldered for a couple of weeks. One good rule of thumb is to tin plate the day you are going to assemble the board. If you do happen to forget and leave a tinned board around for a couple of weeks before assembling it, you can strip the tin oxide with a green potscrubber and abrasive kitchen cleanser powder.



G. Labeling

- 1) Labeling the component side of the PC board will put a "legend" or physical representation of the components in their proper place on the board. This is achieved by doing a 100% (1:1) printout of the "Top Overlay" on the same type of photo gloss paper that we used for the resist step above and then transferring this laser toner pattern onto the top (component) surface in exactly the same manner that we ironed the resist pattern onto the copper side of the board.
- 2) Set all components for Label-Value OFF and all Designations ON. Print the Top Overlay MIRRORED 1:1 onto photo gloss paper.
- 3) After the board has been etched, drilled, and plated, see the instructions for "double sided board" in section C.4.h above and iron-towel the legend onto the top side of the board.
- 4) CAVEAT -- to the extent of the measuring equipment and the types of laser toners that we have used here, we have determined that laser toner is nonconductive. If your double-sided high-impedance or high-frequency circuits are not acting "normally", etch a board without top labeling and see if perhaps your brand of laser toner is not acting like our tests have shown.



H. Project PC Board.

1. Use schematic 6622-5801.ckt (next pages). Using the supplied schematic as a guide and the components as listed below, draw your own schematic. Use the title block to give it your own particular ID (xxxx-5801.ckt, where xxxx is the 4-digit number you have chosen as your own).

Reference Designator	First Selection	Second Selection	Third Selection	Fourth Selection	Notes
C101, C104	Capacitors	Polar	Elecrad	10u	
C102	Capacitors	Polar	Elecaxi	2200u 50 volt	1
C103	Capacitors	Plastic	Mylar	100n	
D101	Transistors & Diodes	Diode	fwbridge	KBP201	
D102, D103	Transistors & Diodes	Diode	rectifir	1N4001	
R101, R110	Resistors	Fixed Carbon Film	QwattV	2.7K	
R102, R109	Resistors	Fixed Carbon Film	QwattV	100K	
R103, R104	Resistors	Fixed Carbon Film	Qwatt	0.5	
R105	Resistors	Fixed Carbon Film	Qwatt	1.5K	
R106	Resistors	Fixed Carbon Film	Qwatt	430	
R107	Resistors	Fixed Carbon Film	1watt	1.0K	
R108	Resistors	Fixed Carbon Film	Qwatt	100	
X101-X113	Connectors	Active	In-Out		2
U102 A/B	Linear Active	IC Amplifier	OpAmp	TL082S	3
U101	Linear Active	Voltage Reg.	vregadj	LM317	4
U103	Linear Active	Voltage Reg.	78XXX	7805	5

Notes:

1. Be sure to use the 50 volt version of this capacitor (there are two 2200 μ f capacitors in the parts bin at the bottom of the Browse column.
2. Place all connectors X101 through X113, then come back and one by one place the "XT" (X Test) version of that same connector pin. It will be necessary for you to double-click the part and change the reference designator manually.
3. U102 is a dual-amplifier in a single package. You will get U102A and U102B when you place the single package.
4. U101 is a power regulator. It will NOT be installed onto the pc board, but mounted to the back of the chassis. However, it will be wired to the board, so we will lay out the PC board as though the regulator will be on the board.
5. Optional. You can buy them at Fry's Electronics.

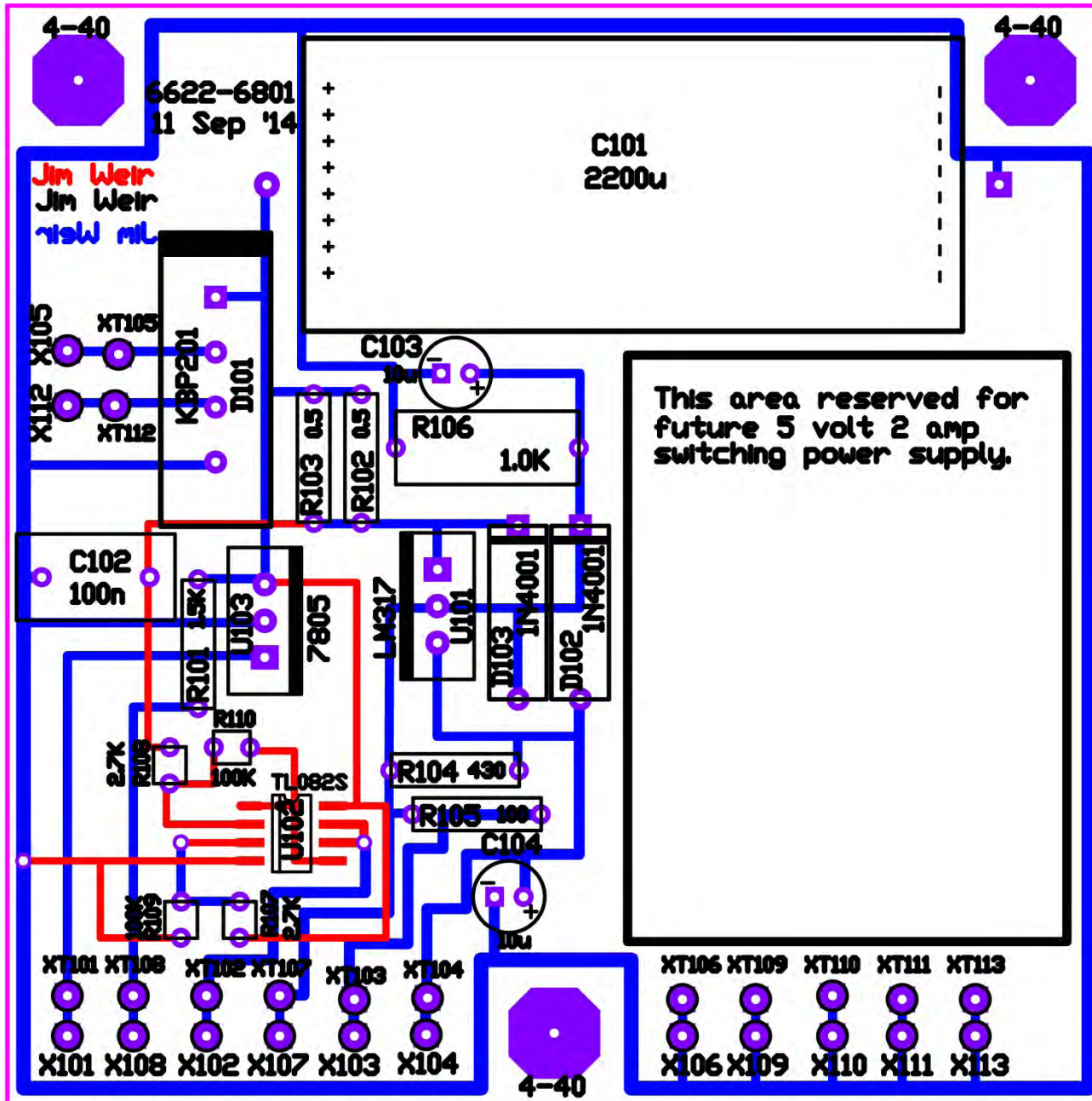
2. Use schematic 6622-5801 shown on the previous page. Using the supplied schematic as a guide and the components in the library "Electronic Fabrication", draw your own schematic. Use the title block to give it your own particular ID (xxxx-5801.sch, where xxxx is the 4-digit number you have chosen as your personal number)
3. Export (File, Export) the netlist. When the netlist screen appears let the screen load Traxmaker with a board size X=3000 Y=3000. Let Traxmaker automatically place the parts.
4. Move the board and parts by dragging a selection box around the entire board and moving the entire board and parts up and to the right about two or three inches.
5. Using the 6622-6801 as a guide (see below), make provision for mounting nuts by placing an OCT 250 pad two tenths of an inch away from each edge in the top two corners and the bottom center (3 places). Use the MEASURE tool to measure center to center distance between the nuts. Cross check this measurement with the mounting holes that are already drilled on your chassis.
6. Run a blue (bottom) trace around the outside of the board as shown. Traces on the BOTTOM of the board are shown in blue and traces on the TOP (component) side are shown in red.
7. Position the parts inside the keep out layer and inside of the blue trace you laid down in step 4. You may make it much easier on yourself later when assembling the board if you spread the parts out fairly consistently across those areas of the board that have no parts.

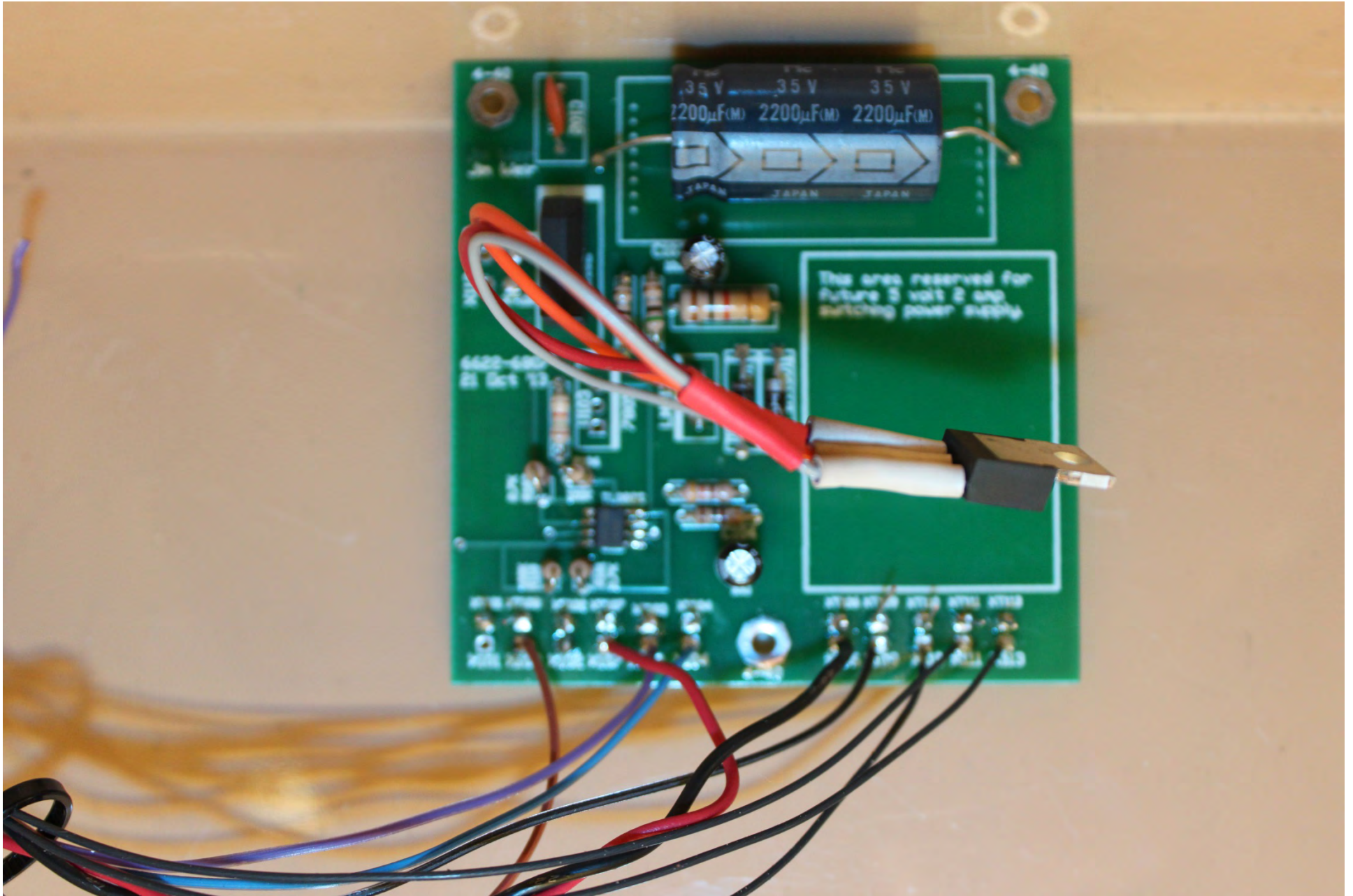
The only part that is somewhat critical for placement is bridge rectifier D101. It should be somewhere along the left edge of the board to mate up with the power transformer leads coming from the chassis (see chassis schematic 6622-2904 on the next page for the chassis diagram).

8. Using the "Nets", "Show Nets", "Show Nets On Selected Pads", select a pad and notice that it tells you what other parts or pads are connected to that pad. Move the parts and pads around until you can easily interconnect the pads with traces. (Note that if you rotate a component, you will have to separately rotate its reference designator so that you can read all the reference designators with either the bottom or the right edge facing down.)
9. Run the traces as the "show nets on selected pads" key tells you to. When you are completed running traces, have the lettering completed, and have the corner shear marks placed, do a "Design Rules Check" with the Netlist, Design Rules Check. Investigate any errors that the DRC routine tells you about
10. If you intend on sending this board out to be made, make the gerbers and nc drill files now using the Traxmaker tools as follows:

- i. File, Create Gerber File
- ii. Default for offsets and undersize zero
- iii. Software arcs 5 mils

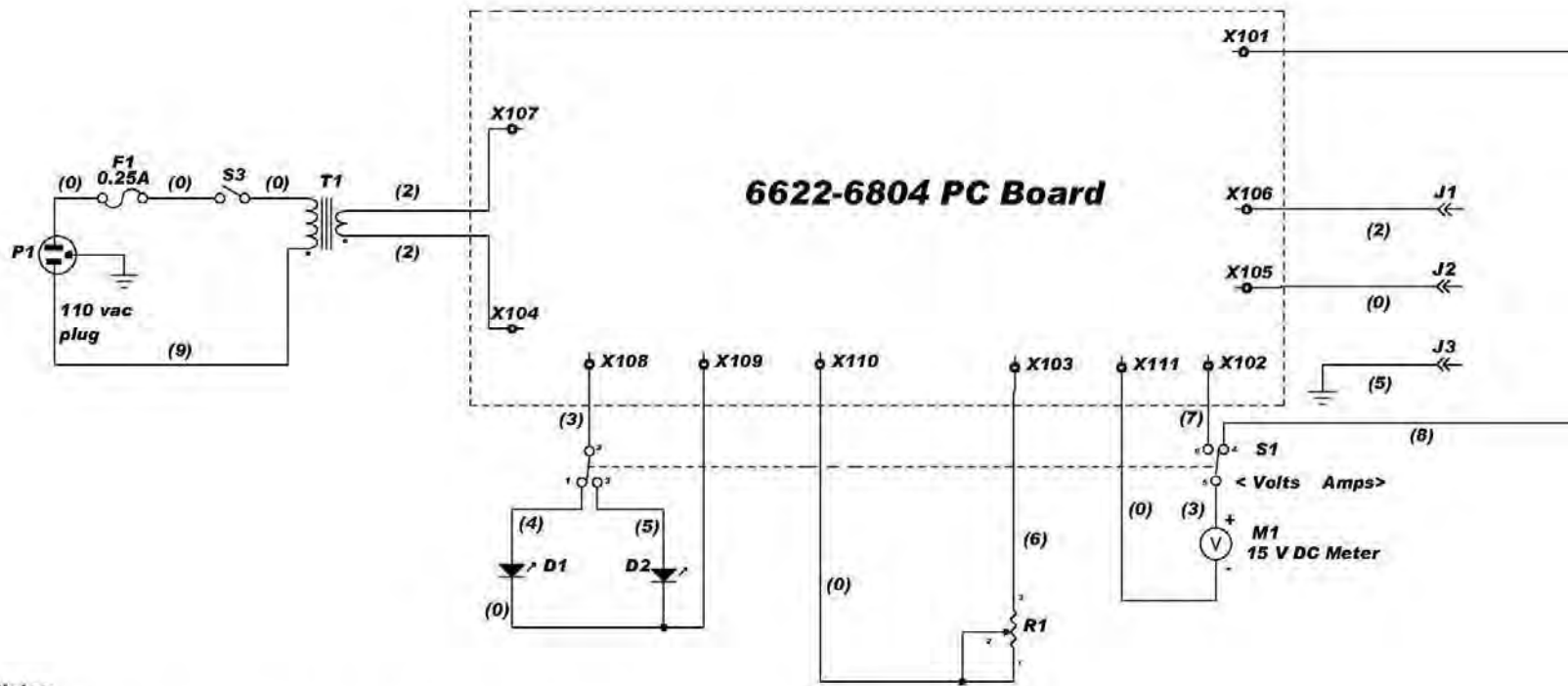
- iv. Autogenerate Aperture File
- v. Embedded Apertures
- vi. More options, check
 - 1. Top Layer, Bottom Layer, Top Overlay, Board Layer, Top Solder Mask, Bottom Solder Mask, Pad Master, Keep Out Layer, include Board Layer, Pads, Text Strings, Path/Filename Layer, other settings Drill Draw 70 mils, Solder Mask 5 mils, Paste and Pwr/Gnd 0 mils
- vii. Accept the default name for the Gerber Filenames (it will save all your gerbers to the same file location as your board artwork).
- viii. File, Create N/C Drill File
- ix. X, Y offset 0 inches, check Create New Tool File.
- x. Accept the default name for the N/C drill files just like the gerbers.





I..Chassis Schematic

1. In addition to a PC Board Schematic, you will need a Chassis Schematic to see how the Board attaches to the components mounted on the chassis (power transformer, voltage control, mode switch,, etc.).
2. (See next page). Here is the Chassis Schematic for the power supply. Some things are worth noting:
 - a. The numbers in (parentheses) next to a line (which are the wires connecting the chassis components together and to the board) are the wire colors in accordance with the electronic color code:
 - i. 0 – Black
 - ii. 1 - Brown
 - iii. 2 – Red
 - iv. 3 – Orange
 - v. 4 – Yellow
 - vi. 5 – Green
 - vii. 6 – Blue
 - viii. 7 – Violet (NOT Purple)
 - ix. 8 – Gray
 - x. 9 – White
 - b. All wires are AWG (American Wire Gauge) 24 stripped from multiconductor computer ribbon cable unless otherwise noted in the assembly instructions.
 - c. In general, wires to the LEFT of power transformer T1 will be AWG 18 and all wires to the RIGHT of T1 will be AWG 24. The exceptions to this are wires to and from J1 and J2 which should be at least AWG 20.



Notes:

1. Numbers in (parentheses) indicate the color of the wire used to make this connection.

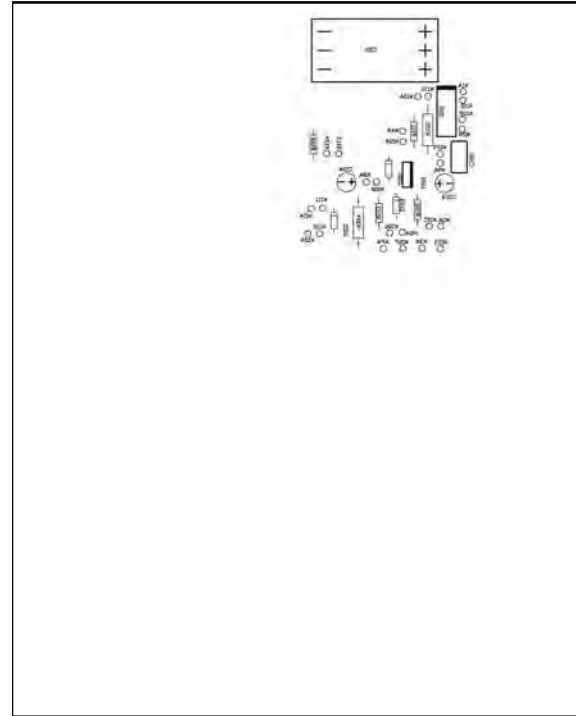
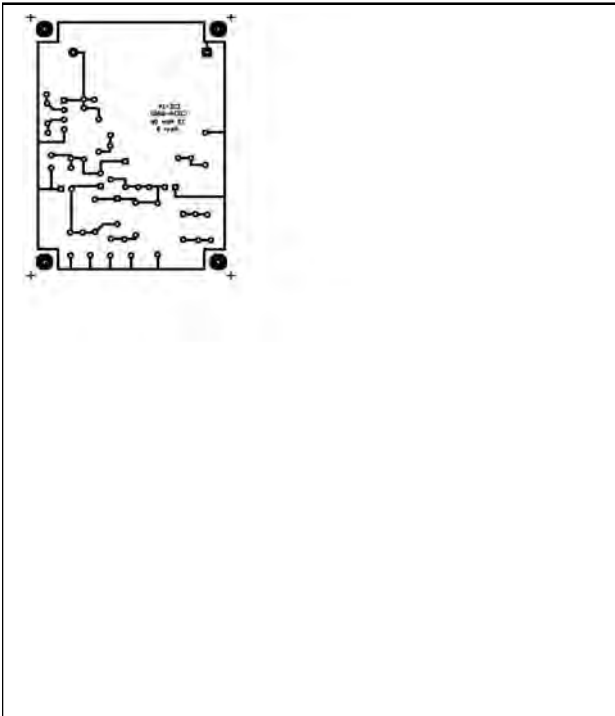
2. All wire #24 AWG unless called out differently in the instructions.

CIE-14 Power Supply Chassis Wiring Diagram	
Rev	ID
A	6622-2904
Date: 8 Sep 08	Page: 1 of 1

J..Making The PC Board

1. Resist

2. If you are going to use the "photo paper - laser printer" method of resist application, print the bottom traces as follows:
 - a. Go to "File", then "Print". Set the "X Scale" to 1.000 and the "Y Scale" to 1.000 . Set both offsets to zero.
 - b. "Position" is Normal, "Quality" is Final.
 - c. Open "Options" and set as follows -- all unchecked except Check Print, Bottom Layer. Include Pads and Single Layer Pad Holes. Other Settings Pad Hole Guide Size 20 mils and the other 3 don't matter. Color mode Black & White. Then OK
3. Make a TEST PRINT on regular paper. It should come out something like this (on the left):



4. Reopen "File", "Print". Leave everything else the same except "Options", Check Print MIRRORED, uncheck Bottom Layer and check Top Overlay. Then OK.
5. Choose an X-Offset of 4.500 inches and reprint **ON THE SAME SHEET**. What should have been added to the sheet is the top overlay components on the right..
6. When you have these two items printed correctly on the same sheet of copy paper, then it is time to show the sheet of paper to the instructor. Once the instructor agrees that this is the correct format, then repeat steps 1-4 above on Photo Gloss (fairly expensive) paper. Wait for another class member to be done with their circuits, turn the photo paper upside down and use the other half of the paper for the other class member.
7. Cut the individual quarter pages with scissors, leaving the most paper margin that you can outside the boundaries of the circuits.
8. Cut a piece of pc board material approximately 4 x 5 inches. Thoroughly clean both sides of the board with a green pot scrubber and copper cleaner. Wash thoroughly with water and dry using paper towels. From this point on through etching, your hands should never touch the top or bottom sides of the pc board.
9. Warm the board **SLIGHTLY** in the 120° F heating oven for no more than two minutes. Place the bottom trace image face down on the pc board and iron the image onto the pc board using the iron on the highest setting. After one minute of ironing, take a doubled-over cloth towel and press the paper onto the surface for half a minute using the heel of your hand for pressure. Repeat the ironing-pressing routine two more times (one minute of ironing and half a minute of pressing).
10. Soak the pc board in warm water until the paper backing starts to peel off of the traces using gentle hand rubbing. It is generally a three step process -- one layer of paper, then another layer of paper, and finally the bottom layer of paper. A 5-minute soak between layers should be adequate in warm water.
11. Remove the board from the water and dry it thoroughly. Inspect the board to see that all the traces transferred from the paper to the board. If there are any traces missing, you will have to repeat this entire section with another piece of pc board material and another set of images. If there are only a couple of minor trace breaks, you can repair the breaks using a fine tip permanent marking pen.

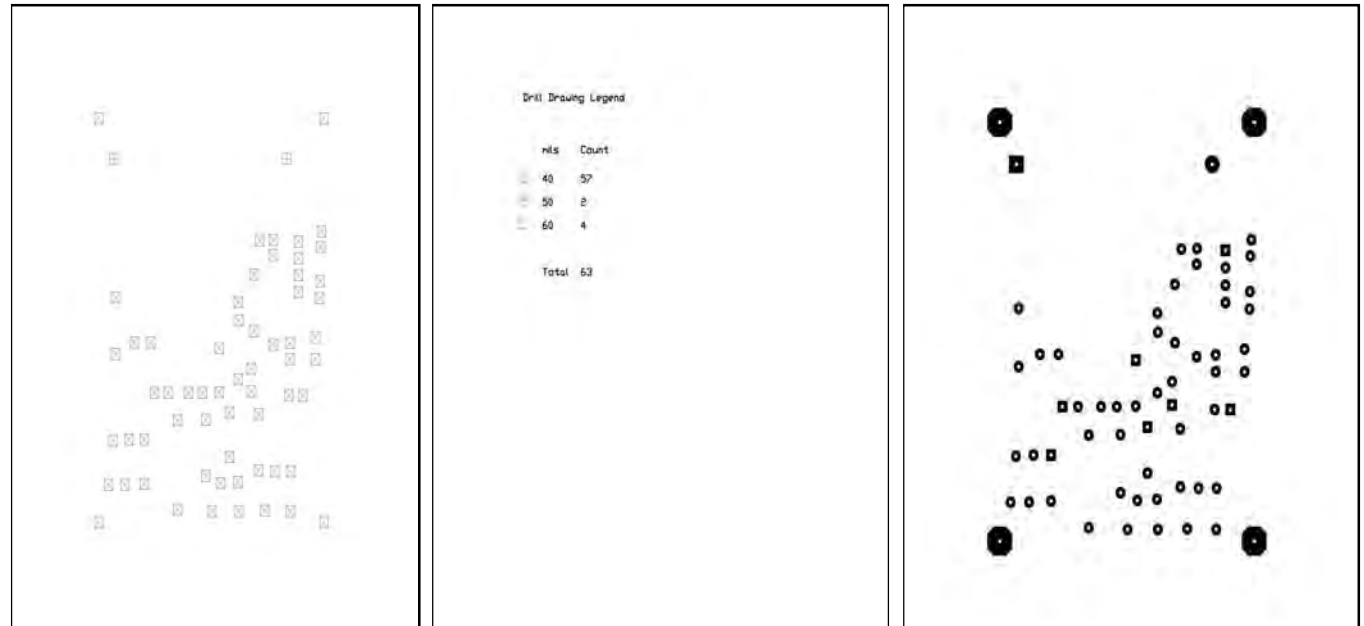
(2) Etching

1. Etch the board in either ferric chloride or copper chloride. If you are using the "homemade" copper chloride method, drill the four corner holes 0.12" in diameter and use nylon standoffs in each corner before etching. (Center punch the holes before drilling.)
2. After etching and thoroughly washing the board, remove the resist using 300-400 grit carborundum paper.

4. Drilling

1. In Traxmaker, File, Print, set the X Scale and Y Scale to 2.000 and the offsets back to zero, Position Centered. In Options, Batch Print, nothing check boxed except Drill Drawing (mirrored), Drill Guide (mirrored), Pads, Vias, Single Layer Pad Holes. Drill Draw Symbol Size 100 mils.
2. You will get three sheets out that will look like this:
3. The sheet on the left is called a "drill drawing". It shows the location of all the holes that need to be drilled. The sheet in the middle is the "drill guide". It shows the size of all the holes that need to be drilled (see note). The sheet on the right is called a "pad master". It shows the shape and location of all the pads on the board.

4. You will note that there are 2 holes that need to be drilled with a 28 mil bit, 26 holes that need to be drilled with a 35 mil bit, 8 with a 43 mil bit, 27 with a 51 mil bit, and 4 that need to drilled with a 125 mil bit .
5. When you are through drilling, wipe the board thoroughly with running water and dry it with a soft towel.



5. Plating

1. Plating in Tinnit® puts a thin coating of tin over the copper. However, the board needs to be thoroughly cleaned and the copper bright and shiny for the tin to plate out. Clean the board with a green potscrubber and copper cleaner and then tin.
2. The tinning should be done NOT MORE THAN 24 HOURS before you begin to assemble the board. The tin forms the oxide with air slowly but surely and tin oxide does not solder without being well soaked in hot flux from the soldering iron. Do not tin one day and attempt to assemble the board next week. It will not solder well at all.

6. Labeling

1. On the COMPONENT (non-copper trace) side of the PC board, take some fairly coarse sandpaper (grit 80-120) and rough up the surface of the board. Wipe any sanding dust from the board with a damp cloth and dry the board thoroughly.
2. Put the board into the warm oven for two minutes.
3. Pick two X-holes (i.e. X105 & X101) that are furthest apart diagonally on the board. Take a scrap piece of wire and place the wire into these holes so that the wire comes out the top (components) side of the board.
4. On the photo paper with the reference designator labels, poke a tiny hole into the holes that you selected in step (3) above. Place the photo paper ink side down onto the board.
5. Iron the legend onto the top of the board in the same manner that you ironed the traces onto the bottom side of the board. Remove the paper backing in warm water in the same way you removed the paper from the trace side of the board.