

DATAK PRINTED CIRCUIT Processes & Instructions

Contents:

Comprehensive Instructions for Datak's different processes for making PC boards. Including a beginners introduction to PC board making.

Introduction:

Datak printed circuit board making processes are intended for two uses: Making a prototype and for small production runs. Users include design engineers, experimenters, as well as students and hobbyists. Instructors are invited to copy these materials freely.

There are a number of methods for producing a printed circuit board described herein. Read the following synopsis to determine which method best suits your projects. Most likely, you will eventually use more than one of the following methods.

The METHODS:

The DIRECT ETCH method is usually the quickest way to produce one small circuit board. All beginners should try this method as it is a good teacher. Not recommended when you need to make many boards or for circuits with numerous components.

Making Positive "artwork"

"Positive artwork" is needed for any of these processes except *Direct Etch*. We suggest that you read this section even if you will be using a computer to produce your *positive*.

Positive Presensitized PC Board Method

This is an ideal method for making several PC boards (or more) and it is quick and accurate as well.

(Positive Continued from column 1)

The method is fast and the most trouble free; getting excellent quality is easy. It is somewhat higher in cost than other methods.

The positive method requires the purchase of boards that are pre-coated with sensitizer. Consequently, you will not have the freedom of using your own PC board stock.

If you have an odd size or shaped board, the negative method may be a better choice.

The Negative Method

This method requires that you spray a negative acting photo resist onto a blank PC board. It does require several added steps (when compared to the positive method). But, you have the freedom of coating any type of PC board material and also cutting and shaping the board before spraying. AND, it is low in cost.

Creating the Negative

To make a board using the Negative Method, you will need to make a negative copy of your PC positive board art. NegFast film makes it easy or use your computer.

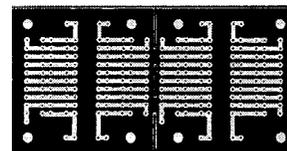
Using Magazine Circuits

Lifting a circuit from a magazine or book is described in this section.

Why make a PC Board??

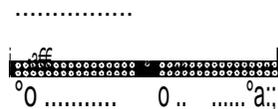
This may sound like a strange question, considering that we are promoting PC board making products here. However, there are times when a PC board is not necessary and may require a lot more time than you want to spend.

A PC board will provide repeated accuracy for any quantity of the same board, but if you are making just one, why not use a Protoboard instead?



Datak Protoboards are pre-etched with copper pads and holes spaced to accommodate standard component leads. All you need to do is place your components, solder jumper wires in place and the circuit is done.

Datak offers several types of Protoboard. The one above is specifically designed for use with DIP IC's (catalog no. 12-607). The Protoboard below is a general purpose pattern with simple copper donuts for lead wires.



This is universal board (catalog number 12-602) with holes and donuts set up on 0.10" centers (as are numerous industry standard components).

These and larger Protoboards are listed in the latest Philmore catalog; they are not yet in the Datak catalogs.

"Think Datak for your Printed Circuit Prototypes"

Comprehensive Datak Instructions for Printed Circuit Prototyping

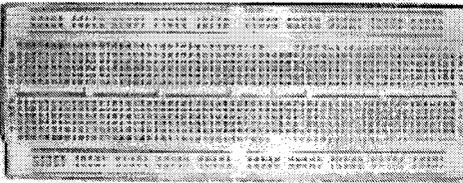
Before you use the soldering iron....

"Breadboarding" a circuit is a term from our grandfather's days... when parts were fastened to a wooden board to see how well a circuit would perform. To provide a quick check to see if a given circuit would work, parts were screwed or nailed to a board. Wires were twisted onto terminals, often never soldered.

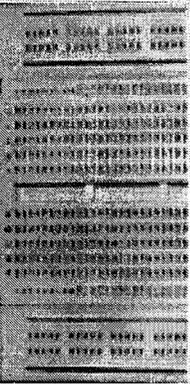
Today we have Solderless Prototype boards for testing circuit ideas. Parts are simply plugged into one of these boards with no permanent connections, such as soldering. The circuit is powered up for testing and circuit changes are quick and easy.

Solderless Prototype boards have groups of holes spaced to accept component leads from IC's, resistors and so on.

Below is a photo of a 7" wide proto-



board.



And here is a blown up photo of the left end of the board. Note the 8 pin DIP IC and resistor. Each pin of the IC is in a hole with a buss strip that has four other holes. You can insert components into these holes or a 22 gauge solid copper wire "jumper".

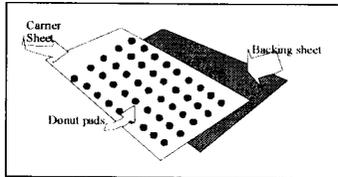
There are several sizes of Protoboards, ranging from small and simple to large with binding posts for power and output (or whatever you want to use them for). A serious experimenter will need at least one board' as it can help you decide if a circuit needs changes; or may not be worth building at all. Check the Philmore (Datak's sister company) catalog for details. **

Methods continued from page one.)

Etching the Board.

Whichever method you use, you will ultimately have to etch the board as the last step before drilling, loading parts and soldering. Etching, of course, is the chemical removal of copper except where your circuit is to be. Datak provides an excellent and economical product for this purpose. The process is described near the end of this booklet. **

Direct Etch, the quickest, cheapest method in detail.



What is a Dry-transfer? It is a sheet of decal patterns, in this case donut pads, on a carrier sheet. The backing sheet protects the patterns during storage. To place a donut from this sheet, you simply remove the backing, place the carrier sheet where you want a donut and burnish with a blunt stick or pencil.

This section describes the use of Dry-transfers in the direct etch process. The Direct Etch method could be used for massive PC boards but we don't recommend it. As a rule of thumb, anything larger than one or two IC's and a transistor (with the usual list of resistors, capacitors etc.) is probably too big. But you decide, after reading the following and maybe trying a small circuit.

The Direct Etch method is simple: Directly on the board's copper surface, you mask the areas that will become your circuit; copper not protected by masking will be removed during the etching process.

The making of a PC board could be described as removing copper from a PC board "blank"; and leaving behind the copper you will want. Before etching the board, you must protect the areas that will become your circuit. Direct Etch is really just a simple, accurate masking process.

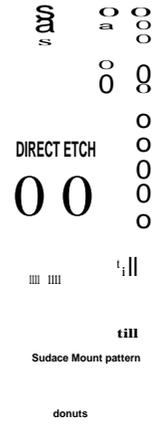
One way to protect the copper would be to simply paint the pads for IC's and resistors etc. with a fine brush and some lacquer or varnish. Actually this is not a bad idea for covering large areas to make a ground plane; the varnish would provide good protection against the etchant. But what an impossible chore for component pads, traces etc.!

This is where Datak products for Direct Etch are a terrific help. Patterns are available for just about any component's footprint. And universal donuts will work if there is not a standard pattern.

You will not need to measure; spacing between component leads (or contacts) will be matched by the appropriate Datak *Dry-Transfer* pattern. And, pre-spaced donut patterns (there are many sizes) can be used widely for "odd", or non-standard components.

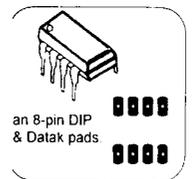
an example:

DIP ("dual inline package") Integrated Circuits have two rows of pins that are 0.30" apart; and the pins on each side are on 0.10" centers. You can easily match this footprint by using any Datak *Dry Transfer* donut pads on 0.10" centers. Simply burnish onto the copper the requisite number of donuts:



Like this>>

Rather than searching through the Datak catalog to find the patterns that you will need, consider



"Think Datak for your Printed Circuit Prototypes"

Comprehensive Datak Instructions for Printed Circuit Prototyping

(Direct Etch continued.)

buying the Direct Etch Master Assortment (no. DE-973). Included are about any pattern you can imagine including surface mount styles.

Resistors, capacitors, diodes and other two-leaded devices are easy; just bur-nish two donuts onto the cop-per.

About the only time that things get a little tricky is when you have a part with no pattern. Then, measure the distance between pins and apply a donut for each lead, spaced appropriately.

Once you have burnished the component patterns onto the bare cop-per, connect the pads either with dry-transfer lines in-cluded in the assort-ment, or use the Etch Resist tape >

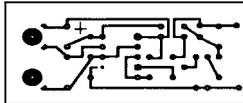
Resist tape >



transfer lines in-cluded in the assort-ment, or use the Etch Resist tape which comes in tiny rolls just like miniature, black masking tape.

Etch Resist tape can be purchase in various sizes. For starters, we suggest two widths, the .031" wide and .015" width. Always use the widest that will fit, it will be easier to handle and a better conductor. With a finger, PRESS tape down firmly; etchant could etch under it when it is a piece loose.

Here is a small circuit with pads connected via tape (shown 112 size).



When the pads and traces have all been applied, you are ready to etch the board as described in the etching section of this booklet. After etching, remove the patterns and traces with a solvent such as acetone, toluene, or finger nail polish re-mover.

Direct Etch tips:

- 1 Use a Printed Circuit Touch-up pen to close gaps or even for traces. The pen's ink will resist etching if you get it on fairly heavily.
- 2 By the way, positive artwork is created in the same fashion, except that you place the patterns on a clear film instead of directly on the copper.
- 3 *Pay close attention to pin numbers; this can be tricky when components are on one side and the copper (or, circuit) on the other.
- 4 When using leaded components, place the components on the side of the board opposite the copper. SMD components have to go on the same side of the copper, of course.

finis

MAKING POSITIVE ARTWORK

"What is a positive?". The term positive does confuse people, but it is simple. A positive of a circuit is the artwork for the circuit done in black on a clear film. The positive (or, positive art) will look like the picture to your right. What you see in black will eventually be copper on the board.

Immediately below the positive is a negative of the same circuit.

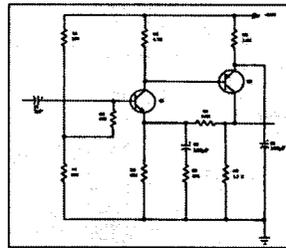
When you use Datak Dry Transfers on clear film, you are creating a positive. OR, when you print a circuit from your computer's software (In most cases; some programs do have the ability to print a negative if you specify it).

In order to make a PC board, whether you plan to use a positive or negative acting etch resist, you will need a positive to begin with. The positive gives you the ability to make many boards from the positive "master" and all will be identical.

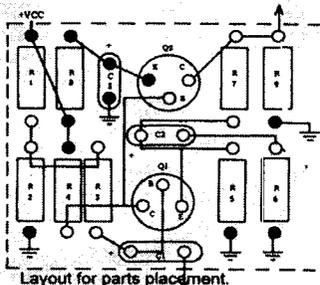
Computer software vs. drafting aids.

There can be no question that the computer is the best way to produce artwork in most cases. If you will be making a multi-layer board, a complex double sided board or simply a very large single sided board, a computer is the answer. If you also require a component side silk screen plus a solder mask, the computer is the answer. HOWEVER, there is definitely a cost in getting started on a computer, which includes a learning curve that may require many extra hours. If you only want to do an occasional board, you may find the "old-fashion" paste-up method much easier. If you are adept at producing a PC layout on a computer, skip the remainder of this section.

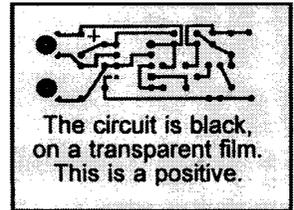
The circuit layout:



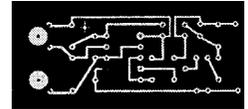
The simple two-transistor amplifier becomes the layout below.



Layout for parts placement.



The circuit is black, on a transparent film. This is a positive.



A negative of a circuit is black-background on a transparent film; that is: the circuit is transparent, clear film.

If you are creating your positive "from scratch", you will first have to determine the layout for the PC board. Having a clear schematic is one thing; converting that schematic into a finished PC board is sometimes quite a trick. You must determine where each component will be located and where each trace will be placed. This text is not going to offer much help to you; circuit layout is such a variable that we must trust to your common sense with only the following tips:

1. Usually, the schematic suggests the PC board's layout; begin with about the same order for major components.
2. Try to stick with a single sided board; its easier in many respects. It is better to add a few wire "jumpers" instead of creating a two-sided PC board.
3. For seriously involved circuits (i.e. large and complex), consider more than one PC board, linked with ribbon cable or jumpers between sections.
4. When you draw the layout, have all components on hand and a caliper for measuring; specification sheets for dimensions are helpful.

(continued)

"Think Datak for your Printed Circuit Prototypes"

Comprehensive Datak Instructions for Printed Circuit Prototyping

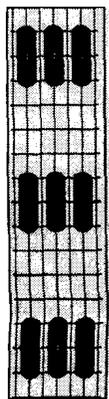
(Making Positive Artwork continued.)

5. Draw the layout on graph paper before committing material to circuit art (see layout above). Where to start? Usually there is a "key" part; an IC or transistor or group of them, that should be the center of the board. After placing them, add the resistors and capacitors etc...

Once you have done your layout....

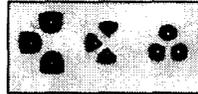
Making the POSITIVE ARTWORK.

The Positive will be "built" on a clear piece of film (like Datak # MS21). You begin this process by placing a grid sheet under the clear film. Then tape the two so that they will not shift. The grid will be a your guide, along with your layout. (Some circuit builders do the layout on a grid sheet and place that under the clear film). Because components are made with leads on 1/10 inch spacing, the grid can help you



with accurate placement of donuts. So, Step- no. 1: Place a grid sheet on your drawing board and tape a clear sheet on top so that the two sheets will stay together. (Use quality masking tape so that it will also come off when finished.)

Step no. 2 is simply to place a donut pad wherever there will be a component lead or pin. We suggest using Datak *dry transfers*, if you place a pad in the wrong spot, just remove by scraping with your hobby knife. Accurate spacing for multi-pin devices such as IC's is accomplished by both the pre-spaced Datak Donut pads and by following your grid. Together, they make the process very simple.

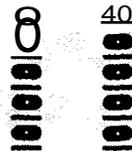


Transistor pads

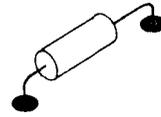
What size donut to use? You don't need to make that decision for transistors and DIP's such as as IC's; just use the Datak pattern made for that part. Note the example; shown are patterns for TO-5, TO-18 and TO-92 transistors; all three leads are transferred together with proper spacing between leads..

But for 1/4 watt resistors and small capacitors, donut size for each lead is mostly up to you. Fairly large donuts are easier to work with but sometimes you may be forced to use very small pads due to space limitations. In general for two lead components, use a donut anywhere from .020" to .035" O.D.. You will soon get a feel for which to use. For parts such as big electrolytics, use a donut with an inside diameter about the same as the components lead wire. Just be sure not to get so close to other pads that you will probably have a solder bridge by accident.

You can even get oval "donuts" (or pads) that have a lead wire between them, which can be very handy in tight places. The oval shape gives a small pad more area for the solder, producing a strong solder joint in a small place. These donut pads are on 1/10" centers, of course, so that they match the spacing of standard IC's and other DIP (dual in-line package) devices.

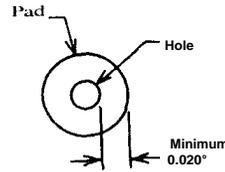


The column to your right depicts some helpful tips; Take your time and shoot for accuracy, checking each trace **twice** When you finish, you are ready to use presensitized positive boards or to make a negative so that you can coat your own PC boards.

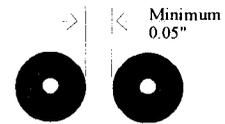


Devices with two leads will be mounted in two donut pads. Spacing is your decision and you may find it convenient to run traces for other components under the body of the part.

Note that we are describing making art for a circuit using components with lead wires, or pins.. That is because surface mount is pretty straight forward (pie) in comparison. After you have done this, you can easily handle SMD (surface mount)

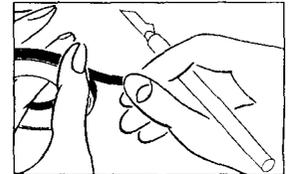


Pad size: Use a size that will surround the lead fairly closely. (Solder will bridge a considerable difference in hole size and lead diameter.) When you can make it fit, select donuts that have a minimum of .020" between the hole and the O.D. (outside diameter).



In general, try to maintain at minimum 0.05" between pads; this reduces solder bridges. (Although some patterns are much closer than this). Try for 0.040" minimum between traces.

The hobby knife, or PC knife, is an indispensable tool that you'll need when making a PC board using any method.



ALWAYS lap tape on top of pad to prevent light leakage during the exposure process (see sketches).



POLYESTER FILM PADS

TAPE

Be sure to lap tape on top of pads.

Finish the artwork by connecting together all pads that need to be connected. When you cannot make a connection between two pads that must be connected because of interfering traces, consider using a jumper wire. (Or, could you reroute interfering traces?) Hopefully, most problems were solved during the layout process.

Once you have finished your Positive Artwork, you can either use it with the presensitized, positive acting boards or you can create a negative for use with the Negative acting sensitizer. Both methods are described in detail in this booklet.

"Think Datak for your Printed Circuit Prototypes"

Instructions for using POSITIVE, PRESENSITIZED PC BOARDS

ABOUT THE DARKROOM.

A darkroom is not needed for Datak processes; subdued incandescent room lighting is generally good. Do eliminate sunlight from windows and, preferably, all fluorescent lights. We suggest either low wattage incandescent *lights* (25watts) at a distance of several feet or yellow bug lights, still kept at a distance. This whenever handling light sensitive products which includes Positive Boards, Neg-Fast film or when applying or handling boards that you have coated with Datak's negative resist.

if you already have a fully equipped dark room, there is certainly no: harm in using it and no problem with safe lights etc..

The WORKING AREA and equipment.

It is important have a good work area for the chemical processes used to make PC boards. It is necessary both to be successful and for your personal safety. You will need the following for certain:

A sink **with** both hot and cold running water; with a faucet that will **MIX** for warm as well. This should be in an area that won't mind some stains. The etchant, in particular, can be messy and stains are likely.

You don't need a darkroom, but you will need to be able to close out sunlight (see above).

Ventilation is important, during etching and when using the negative developer. In a confined area these chemicals may cause throat and eye irritation and even lightheadedness even sickness could result. Providing an air **flow** should prevent any reaction for most people; use a fan to exhaust fumes' via a window.

Goggles are a **very** good idea when working with chemicals; and being close to a sink in case an eye; needs to be rinsed out.

ALWAYS keep the chemicals out of reach of young children. Ingestion can be very harmful or fatal.

POSITIVE PRESENSITIZED PC BOARDS

This is the easiest to use of the various methods, and is generally the most accurate. Although the negative method may cost you less money, the time savings and fewer steps might compensate for material costs being a bit higher.

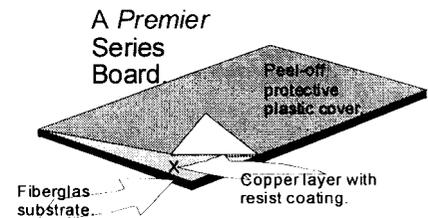
What are they? The boards are conventional, 1 oz. copper PC boards with a Fiberglass substrate. The copper is coated with a light sensitive chemical that is often called a resist or a sensitizer. When this coating is exposed to enough ultraviolet light, it will rinse away in the developer solution, exposing the copper beneath. The copper is then etched.

In brief, these are the steps we are about to describe in detail:

1. Expose the PC board with your positive artwork on top of it to UV light.
2. Develop the board in liquid developer.
3. Etching the PC board will be covered in the section about etching.

Before we start with the procedure, we need to advise you that there are two different types of positive board. They are very similar in many respects, but you will want to be aware of the difference. The *Standard* series has been around for many years. The catalog numbers for the *Standard* series always begin with a "12" (e.g. 12-024 etc.).

The Premier series is a deluxe version and a newer innovation. It can be exposed with a photoflood, or even a standard incandescent 100 watt light bulb, uses less developer and has a protective plastic, peel-off film that is a big plus. The resist coating is also superior in that small imperfections have been reduced to almost nonexistent (very rarely needing touchup after developing). The Premier series is somewhat more costly (average 15-20%). Premier boards have part numbers beginning with the numeral "14" (e.g. 14-024 etc.)



The two biggest advantages in using the Premier series are the higher quality coating and the plastic film protective cover. While this peel-off cover is in place, you can measure, mark and cut the board to size in normal room lighting, even bright lighting. Standard boards must always be handled in very subdued light.

Confusing Terminology. The industry tends to use several terms for "Positive-acting PC boards". A positive board is a positive board and do not be distracted by variations such as "positive acting", "pre-sensitized" or coated etc.. And the terms *Precoated* and *Presensitized* are interchangeable as are resist and sensitizer. Technically, the coating is a resist and not a sensitizer, the resist is light sensitive. The term "positive" is the giveaway

However, Standard boards should not be considered to be terribly inferior; they have been the industry's mainstay for nearly two decades and can be counted upon for very good results. Here are the main differences

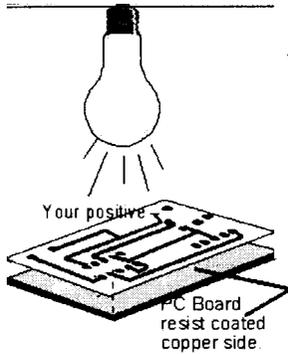
>Standard boards require UV light for exposure, Premier boards do not.

>Standard boards, mix developer with three parts water vs. ten parts water for Premier boards.

>Standard boards should be checked after developing for possible touchup before etching; they may need just a dab with your PC ink pen.

(Continued next page)

Instructions for using POSITIVE, PRESENSITIZED PC BOARDS.



Your positive will be placed on top of the resist coating on the copper side of the PC board during exposure. Be sure that the positive is not upside-down; the circuit in copper is going to be oriented the same way as your positive on top of the board.

Step 4. Expose the board with your positive. Follow the Exposure Chart for exposure time. Use an exposure frame.

Place your positive on top of the sensitized coating on the PC board and sandwich the two in the exposure frame.

The Exposure Frame: This device will be familiar to you if you have ever made photographic prints. The frame is simply a holder for a small board and piece of glass. With your positive in place on top of a precoated board, put the two in the frame under the glass and slide into the frame. The frame will hold the two tightly together for two reasons: (1.) the positive should never move around during exposure and (2.) the positive is pressed tightly against the board so that light does not get under the positive. Now expose to light.

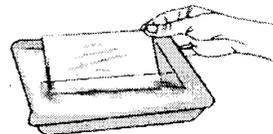
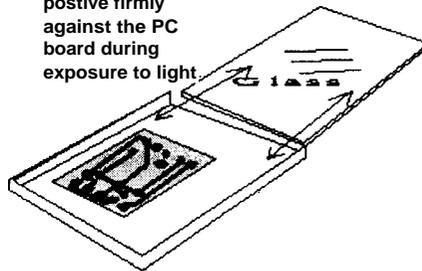
The Light Source: See the chart at the bottom of this page. We are assuming that your line voltage is between 115 and 125 Volts AC .. if below this range, you may have problems. Also, for the sun, keep in mind that its ultraviolet output could vary; especially in large cities, on hazy days etc. .

Note that the Premier series can use an ordinary 100 watt white incandescent light bulb.

Develop the board: While the board is being exposed, prepare a tray of developer solution. Mix the Datak developer concentrate 1:10 with water for the Premier series; or 1:3 for the *standard* series. NOTE carefully; mix with very warm water. A cold developer is not going to work, it should be above room temperature or warmer. (Be sure to use only Datak nos. 12-402 or 12-406.)

Use glass/plastic photographic tray. Use just enough developer to cover the board. Rock the tray gently until the copper is fully exposed in areas to be etched. It will be obvious to you when the copper is bare; be sure that there is no haze or film remaining. Rinse in cold water for a couple of minutes. (Continued next page.)

The exposure frame; holds the positive firmly against the PC board during exposure to light



TINNIT® Tin Plating for PC boards. Kit no. 12-800

This is a slick product that should be of particular interest to professional users. When you get to the point where you have a finished PC board, you may want to tin plate the copper for two or three reasons.

The tin plate, which solders very easily, looks very sharp, shiny and clean. If you are making a board for a customer, tin looks quite professional. And the look lasts.

TINNIT will also provide protection for boards that will be stored for later use; they will remain solderable and the copper will be protected.

INSTRUCTIONS
TINNIT is electroless, requiring no electrodes or wires.

NOTE: WE RECOMMEND GOGGLES and RUBBER GLOVES WHILE WORKING WITH THIS PRODUCT.

1. Dissolve the TINNIT powdered concentrate (marked NO. ER-18) in 12 ounces of very warm water (temperature about 130° F) and then add water to make one pint (16oz.).

NOTE: Use only a plastic or glass container mixing and storage. Do not use metal containers or caps with metallic linings.

2. Prepare the circuit board for plating by removing all traces of resist with either acetone, lacquer thinner or DATAKOAT® thinner. If oxides are present, clean with scouring powder rather than soap or detergent; do not use steel wool.

Continued on page seven)

Board T type	F15 T 8-BL X2 bulbs**	app. noon SUN*	Datak ER-39 bulb	mix developer concentrate	100 watt bulb Premier onl
STANDARD R	12 to 14 min. @ 7 inches	8 to 10 min. any distance	12 to 15 min. @12 min.	3 parts water to 1	not applicable
PREMIER	9 to 12 min. @7"	4 to 4.5 min.	10 min. 12" distance	10 parts water to 1	10-12minutes @12" distance.

** look for deskclamps that hold two bulbs

*sun about noon to two P.M.

Etching: You are now ready to etch the board; see the section on etching later in this booklet. After etching, you will be ready to drill the board and mount the parts, solder and finish the circuit.

Final notes: Here are some helpful tips for users of the positive PC boards:

1. Store used developer in a plastic or glass bottle. Plastic is best as you may squeeze most of the air out of the bottle and cap tightly. Do this because the developer will react, over time, with the carbon dioxide in the air and decompose.
2. You can use the developer until it simply does not perform; then flush it down the sink with a fast cold water chaser. Storing developer in a refrigerator can help to keep it fresh for very long periods.
3. We suggest leaving the resist on the board until after drilling. Remove the resist with solvents including acetone or a non-acetone fingernail polish remover. The later is available at most drug stores and discount stores.
4. IF the board proves to be tough to etch, you probably did not develop it long enough and some resist residue remained behind. Or, said residue can also be caused by too short an exposure time, too weak a light source and so on.

ooo

Making a Negative from a positive or a Magazine page circuit.

Neg-Fast film is a product that will help you to easily create a negative for use with the Negative Photo Resist Spray from Datak or any other negative acting resist. *Neg-Fast* is a plastic sheet, about as thick as a sheet of paper. It is available in small sheets 5" X 6" or larger sheets 11" X 8". When exposed to light with your positive and then developed in a special liquid, a negative is produced. The process is simple and very reliable.

INSTRUCTIONS:

Handle *Neg-Fast* film in very subdued light and with no sunlight or fluorescent light present. It is also OK to use yellow light.

For use with a magazine article/circuit:

Step 1a. IF the circuit has no printing on the reverse side of the page, you can clip it from the magazine. Or, you can use an office photocopier and make a copy. Make a good, high contrast copy with the circuit plenty dark, other areas white.

Step 2a. A printed magazine clipping or photocopy such as this requires an exposure time of four minutes at a distance of 24" using the Datak no. ER-39 bulb. Except for this exposure time, follow the steps as given below.

For use with a pasted up or photographic Positive.

(This is a positive circuit on a transparent film.)

Step 1. Use scissors or a sharp PC knife, cut the film to a size matching your circuit positive (or a bit larger).

Step 2. Place the film in an exposure frame with the emulsion side up*. the positive on top of the film (the positive can aceither direction, there is no "right-side-up".) Close frame securely.

(Continued next page)

(continued from page six.)

Remove all cleanser residue by washing thoroughly in plenty of clean water: If you do not thoroughly remove all cleanser and etch residues, they can seriously deplete the TINNIT bath.

3. Place the TINNIT solution in a Pyrex tray and heat to between 120 and 140 degrees F. Immerse the board for between 10 and 30 minutes. Agitate for a moment at five minute intervals. A bright tin plating will result.

4. Rinse the board with running tap water and dry with paper towels.

Special notes:

If the TINNIT solution is left standing it may evaporate, leaving a precipitate (white or light yellow). The precipitate will again dissolve when water is added and heated and is perfectly usable.

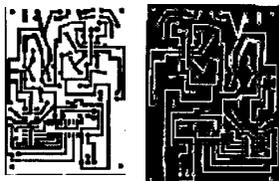
Do not leave, powder residue to heat on stove; heated dry salts can decompose and release very irritating sulfur dioxide and ammonia gases.

TINNIT may be stored for about six months (longer if refrigerated) in an airtight bottle made of either plastic or glass. Avoid metal bottle caps.

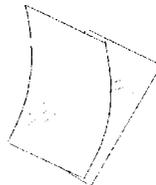
TINNIT WARNING: This product contains acidic tin salts. Although it contains no cyanides or other highly poisonous materials, it is a skin and eye irritant. In case of eye contact, flush immediately with running water. Consult a physician as soon as possible.

IF ingested, do not induce vomiting. Call a physician. Drink plenty of water, as much as possible.

Never leave these chemicals in the reach of children. Younger teenagers should be supervised by a knowledgeable and responsible adult.



A positive on the left, a negative on the right.



(continued from page seven)

Step 3. Expose to the Datak ER-39 bulb at a distance of 24" for 1-1/2 minutes (4 minutes if a paper positive).

Special note: For circuits larger than 4" X 4", you may find the outer comers not fully exposed. For larger boards, you may suspend the **ER-39 bulb** so that it swings freely on its power cord. During a five minute exposure time, keep the **bulb** gently swinging **randomly back and forth** across the board which is in its exposure frame about 24" directly below the bulb. Use a porcelain socket as it can take the heat.

Step 4. Use only as much developer as needed; do not return used developer to the bottle. Mix the developer concentrate with four parts of very warm tap water. Place the developer in a shallow tray (either glass or plastic, not metal). Develop for about two minutes; or until you can see that you have a good clear image of the circuit's negative. The developer's temperature should be warm, not less than about 72 degrees F but not above about 90 degrees. When fully developed, gently rinse in lukewarm tap water.

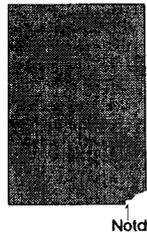
Step 5. Mix one part of the Fixer concentrate with 4 parts of warm tap water and allow the negative to stand in this solution for about five minutes. (If you are using the developer tray, be sure to rinse it out with water first.) It is necessary to use only enough to cover the film but you may return unused fixer to the bottle. (When it comes to the time to buy new developers, throw away this old fixer as the developer package comes with new fixer too.)

Step 6. Rinse again in fresh, warm tap water and place gently between a couple of paper towels to absorb excess moisture. You may then dry by leaving it on top of another, fresh paper towel or hang it with a film clip. You may use warm air or heat to hurry the process, but do not exceed 122 degrees Fahrenheit or you *will* distort the film.

Step 7. Touch up may be done with a black pen such as Datak nos. ER-15 or extra fine point ER-25. Eliminate any pin hole spots, if any.

Step 8. Finished. Use the negative with a PC board coated with Datak no. ER-71 sensitizer or with other maker's negative-acting PC boards.

Exhausted fixer and developer may be flushed down the drain; follow up with a couple of minutes of fast running cold water.

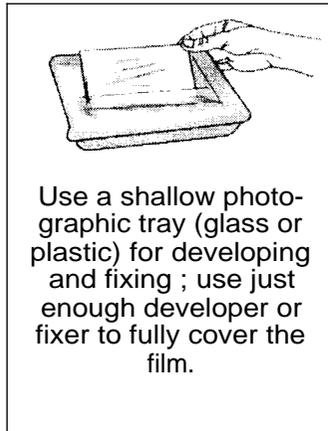


Notch

Identifying the emulsion side of the film.

Each sheet of film will have a notch along one edge. Hold the sheet up in front of you; when the notch is on the bottom

edge and closest to your right hand, the emulsion side is facing you. OR, simply scratch each side with your PC knife tip. The emulsion side will scratch away revealing a clear film; the non-emulsion side will not show scratches.



Use a shallow photographic tray (glass or plastic) for developing and fixing; use just enough developer or fixer to fully cover the film.

TIPS & MISCELLANY

Computer Software ... if you are considering a computer program for drafting your PC board layouts, PCDATAK may be a good choice. It is a DOS program but can run under Windows or Windows95. It is very low cost, easy and quick to learn and incredibly flexible for a low cost program. To print circuits, you need to have a LaserJet printer with the Postscript feature. Or, take the circuit on diskette to Kinko's as they have such printers. Datak's catalog number for the software is 12-077.

Soldering ... PC circuit layout descriptions in this booklet are assuming that you will be hand soldering your final circuit board. If your prototype will eventually be a production item and you are planning to use dip or wave soldering equipment, then circuit layout needs to be done accordingly. This booklet does not go into enough depth to teach techniques for true production boards.

For production, use a software to do your layout. Most programs incorporate accepted techniques that will keep you out of trouble. Datak's software, PCDATAK no. 12-077 automatically provides correct spacing between traces, donuts etc..

Drilling the PC board ... drilling with a standard, household hand drill can be an excruciating chore; and it is very difficult to do a decent job. We suggest that you use only a drill press, even if you have to beg a neighbor to use his. If you use a hi-speed mini-drill such as those from Dremel, use a light dimmer to slow it down about 40%. (You will get a feel for the right speed after using it for a short time.) A slowed-down Dremel with a Dremel drill press' is the ideal way to drill your boards.

Small bits are available from Datak and also from most industrial **supply** houses which will carry dozens of sizes.

NEGATIVE PHOTO RESIST for PC BOARD MAKING

The Negative Acting Photo Resist method

The method is based upon using the Datak no. ER-71 Photo Resist liquid, which is a light sensitive spray-on resist. The small, 4 ounce bottle will cover about 1,700 square inches and is applied with the pump action sprayer included in the package. Good results can be expected if you carefully follow the steps.

You will need to begin with a quality negative. If you have a positive, see the section on using NegFast film to produce a negative. Or, use standard photographic process to make a high contrast negative.

Step 1. Clean the PC board. This is a critical step and must be done well. Scour the board with water and a common kitchen cleanser (containing bleach) and very fine steel wool. Scrub the board and rinse until water sheets out evenly over the entire surface when you hold the board horizontally under a pencil-lead stream of water. If the water tends to pull away from the boards edges or any point on the aboard, clean it again. If the board cannot pass this water test, it is "dirty" and must be cleaned until it passes.

Dry the cleaned board with paper towels. You could wipe with ammonia (never with alcohol). Handle by the edges, don't touch the surface.

Step 2. Coat the Board with Resist. Do in subdued light* The board should be warm, ideally about 120 degrees F. Heat the board on a hot plate or you could warm it with a heat gun or hair dryer.

Lay the board flat and spray a light, continuous film. (Sink marks will indicate poor board cleaning.) Sight along the board surface toward your light; make the coat as even as possible.

Drying the board in a dark cupboard works fine if there is plenty of volume in the cupboard. But if the volume is quite small, there is **sometimes not enough air to absorb the fumes and the board takes a very long time to dry.** Don't expect a small drawer to dry two or three boards overnight.

Step 3. Drying the PC Board. The board may simply be place in a dark cupboard to dry over night. Or, bake it in an oven at 140 de grees F for twenty minutes; the oven must be dark. Allow the board to cool ten minutes before use.

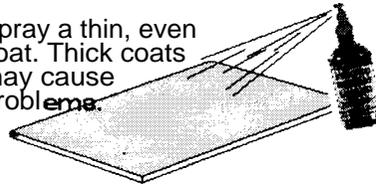
If your coating is not a clean looking, even coat with few signs of bubbles, consider repeating the process by adding a second coat. Strive for a thin coat, and as even as you can make it.

Step 4. Exposing the PC Board. Place your negative on top of the resist coating and place the two in a contact *exposure frame*. The frame will hold the negative and board tightly together to prevent light from leaking between them.

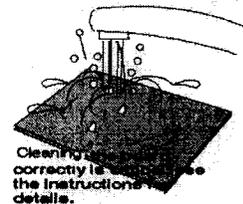
Expose the board to light using one of the following choices:

* *subdued* light. After you clean the board and until you finish the developing stage, work only in subdued lighting such as a 25 watt bulb several feet away or a yellow "bug" light bulb.

Spray a thin, even coat. Thick coats may cause problems.



Sometimes steel wool will contain oil; but it can be removed with some soap and running water. Soap, squeeze and rinse a pad two or three cycles and it will be **clean** enough to use. A liquid dish soap is suggested.

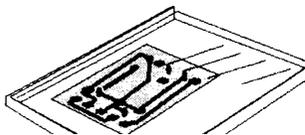


Spraying the board is an important step; try to get a thin and even coat. A thick coat can result in ragged edges on pads and traces in your finished board.

tip: Pump the spray fully, do not try to squirt a light coat by pumping with short strokes. Start spraying off **the edge of the board and sweep** across the board; each pump cycle should be a full stroke. **(Short strokes tend to shoot globs, which are not good.)**

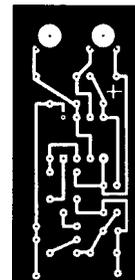
Always spray on a warm board; a board that is cool is very tough to coat properly. **Some users let the board dry and apply** a second coating.

The negative is clear (transparent) where there will be circuit and the "background" is opaque.

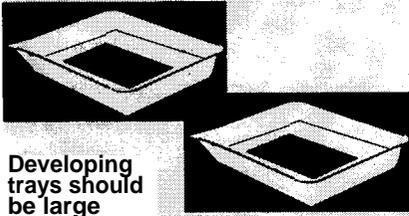


The exposure frame clamps the positive or negative tightly against the PC board; and the glass top allows exposure to light.

You can make a negative using Datak Neg-Fast film or photographically. Also, some software will make a negative of a PC board layout.



(Continued on next page)



Developing trays should be large enough that the PC board can lay flat in the bottom.

(Negative Photo Resist continued from previous page.)

- 1.) Sunlight at about noon, for three minutes.
- 2.) Datak no. ER-39 bulb, for 5-1/2 minutes at a distance of 20".
- 3.) Two F15T8BL bulbs 20 minutes at 15".

THE WORK AREA SHOULD BE WELL VENTILATED; AVOID BREATHING VAPORS and avoid skin contact. Do not use near open flames or sparks.

Step 5. Developing the PC Board...

You need two metal or glass photographic trays for developing (never use plastic trays). If you work with small boards, find some smaller trays which will save on developer.

Use only Datak developer no. ER-8. Place enough developer in each tray so that the board is under the developer by about 1/16" when it is placed flat in the tray.

Note: The resist pattern is transparent during developing and cannot be seen until it is completely dry.

Follow these developing steps:

- a.) Place exposed board in first tray for 30 to 45 seconds, gently rocking the tray every five seconds. (Never touch the surface of the board, the resist will scratch easily. Handle by the edges.)
- b.) Remove board, holding by the edges. Allow it to drain for ten seconds and then place it face down on 2 or 3 layers of clean facial tissue (don't slide or rub!).
- c.) Carefully remove tissue and immerse the board in second tray for ten seconds. Remove, allow it to dry and inspect the pattern against the light (hold board flat and point it toward the light; sight along board). Good developing shows up the pattern clearly with no drain marks or stains in the areas to be etched. Old, used developer will leave globs at random.

Dry thoroughly before etching; at least ten minutes at room temperature. The resist must be hard before etching.

Step 6. Touch up. The board may now be handled in normal lighting. Inspect the board for breaks in traces or pads. If there are any, you can touch them up with the etch resist pen. Repair any breaks before etching. If the above steps were done well, touch up will probably not be needed.

Etch the PC board. (See section about etching.)

OO*

The Products...

The no. ER-71 Photo Resist (left) is supplied in a dark bottle which will prevent deterioration. Officially, the shelf life is about two years; but the actual life is probably much longer. It can be stored at room temperature; cooler temperatures will add to the shelf life. Warm it up to at least room temperature to apply it. The Resist has a spray cap for application. Spin coating is also good if you are equipped to do it.



The no. ER-8 developer is supplied in a metal can and will keep for a long time if kept tightly shut. If you plan to do quite a number of boards, then you'll want to start with two cans. The Resist will cover about 1,700 square inches of copper but a can of developer won't develop nearly that much area. (Area will vary with circuit density.)

Ragged edges on the resist image mean that the resist was not completely dry before the exposure, OR, that the resist was applied too heavily and should not have been such a thick coating.

Spread-out lines occur if you exposed too long or if the negative was not in perfect contact with the PC board. This can happen if you use a warped board. If the pattern is bad, you may as well salvage the board by cleaning off the resist with lacquer thinner and cleaning/recoating. (A warped board can usually be gently bent into shape.)

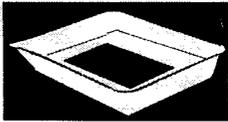
The Datak ER-8 Developer contains aromatic, partially chlorinated hydrocarbons. The product can attack plastics and may discolor vinyl counter tops, attack wood finishes and so on. In addition to the precautions listed on the package, don't use this product in areas that can't stand some abuse.

Never leave such chemicals where children might get into them. Ingestion of this and other PC board making chemicals can be very harmful and even fatal, especially in small children.

Fast removal of all manner of etch resists can be done with an acetone free fingernail polish remover. It can be found near the fingernail polish in drug stores and discount stores etc.. It is a cheap and effective way to go.

ETCHING PC BOARDS

Ferric Chloride etchant can stain clothing (and just about anything else) horribly. Wear old cloths and don't use this in an area that can't take some stains. Work close to running water; an immediate cold water flush can reduce staining, rinse out an eye etc..



Etching can be done in an open, photographic tray. If you keep the tray covered with a piece of window glass when not in use, it will reduce the fumes and can be kept this way for many hours.

NEVER add water to the Etch Powder Concentrate!
Add the powder to water.

HEATING &
Boiling !!



The mixing of water with Ferric Chloride powder generates considerable heat. You must add the powder to the water; NEVER add water to the powder!

Used ferric chloride etchant may be stored in plastic bottles for future use. If you will be storing it for quite awhile and the bottle is not full, squeeze the bottle to eliminate most of the air space. It should last many months.



The ETCHING PROCESS.

There are any number of chemicals that may be used as an etchant. The only one that we recommend is Ferric Chloride. It is available from some sources as a liquid and Datak supplies it in powder form to be mixed by the user with water. This is the most effective and economical of the etchants.

Factors that will accelerate the ferric chloride etching process are heat, oxygen and agitation. The etchant will work at room temperature but at higher temperatures (say, about 125 degrees F) it will work much faster and cleaner. Agitation not only keeps moving fresh developer over the copper, but introduces oxygen, which acts something like a catalyst.

Ferric Chloride is not an acid; the chemical process is one of displacement. We do not recommend acids for many reasons of safety and effectiveness, disposal problems etc..

The etching container must be either glass or plastic. It can be as simple as a photographic tray (that you keep rocking gently) to the Datak etching tank, which bubbles air through the solution to provide both agitation and oxygen. A fish-tank type heater adds to the effectiveness of the etch tank. Agitation is very important and proper etching won't happen without fairly constant of the etchant during the entire process.

Using Datak Ferric Chloride We always recommend that you wear eye protection and provide **adequate** ventilation. The following assumes that you will use an open, photographic tray. Mixing is the same regardless of what devices you use during etching, i.e. the etching tank, tray etc..

Step 1. Mix a package of Datak number ER-3 ferric chloride powder with 16 ounces of water. Add the powder slowly; expect heating of the water as you go. Adding too fast can cause splattering due to the fast heating. An empty, plastic one gallon milk jug works fine for the process. Mix until all powder is dissolved. Then store the finished product in a plastic bottle. It will keep for a long time.

Step 2. Preheat the etchant. You could preheat the etchant by immersing its storage bottle in a bucket of hot water for twenty minutes before pouring it into a tray. IT WILL EXPAND WHEN HEATED, LOOSEN THE BOTTLE CAP SO THE BOTTLE WON'T BURST. Or you could use a Pyrex glass tray on a low-heat hot plate (etc.). A heat lamp could be used on an open tray. Room temperature will work, but slowly.

Step 3. Etching can take from ten to twenty-five minutes. Immerse the PC board and continue agitation by rocking the tray gently, or lifting and immersing the board with a plastic tweezers etc..

Step 4. Inspect the board periodically and it will be obvious when etching is complete. When finished, rinse the board with fresh water, wipe off with paper towels. The process is completed and you are ready to drill the board, remove the resist and load the parts.

You may pour used etchant back into the container to use it again. Replace the etchant when the process slows down considerably. Old etchant may be flushed down the sink but be sure to follow-up with a vigorous water chaser for two or three minutes. The etchant will dilute and disperse quickly.

Users note: Ferric Chloride etchant is available from several sources. Some is not of good quality and you will note that it is slow to act and etches only a very small volume of copper. Datak provides a very high quality etchant and at a price that is better than some of the lower grade materials. We do strongly urge you to use the Datak product. Slow etching can cause undercutting, weakened resist and other problems.