Producing PCBs from positive artwork and positive resist PCB material

Producing PCBs from positive artwork and positive resist PCB material

February 2010

Beware - the chemicals used in this process are poisonous and must be handled in accordance with the individual manufacturer's instructions.

Note that ammonium persulphate **can** be used to etch the copper laminate on PCB material created using most other methods (eg iron-on...) --- **however** --- RS Components warns against using this chemical on the Fotomechanix / Fotoboard2 series of products, the ones specified below in step 3. They specify ferric chloride as the etchant - take note of this if you intend using this PCB materials range from RS.

Local copy of Fotomechanix PDF (/~vk4adc/web/images/UserFiles/File/pcbs/fotomechanix info.pdf) & FR4 PDF (/~vk4adc/web/images/UserFiles/File/pcbs/fotoboard FR4.pdf) files

If you have visited other pages on this website, you will have noted that many of my electronics-style projects are based around building up the actual circuits on printed circuit boards. I started doing my first PCBs back in the 1970's when I first drew the 'artwork' directly onto the copper cladding with a 'etch resist pen' - these days often called a 'Dalo pen' after one of the original pen manufacturers. In the years between then and today, I have used several other methods including negative and positive resist pre-coated and self-coated material. My circuit board artwork often took me weeks to create part-time with donut pads and black adhesive tapes on clear acetate sheets, then cross-checked a number of times to make sure the components fit in the spaces and there weren't any layout errors. It was a time consuming process and wasn't exactly cheap.

These days the process is a lot simpler and almost anybody can create printed circuit boards for most of their projects. It still requires patience, accuracy and practice to develop a schematic out into an actual PCB layout but the tools are available to almost anyone with a normal PC in their home plus a few assorted plastic containers for the developing and etching fluids to be put in.

At this point in time, read the whole web page to see what the each step is and what materials (PCB material, developer, etchant, containers ...) you will require. Later you can work through the relevant sections step-by-step as you actually produce your first PCB.

Some of the images will expand on your screen as you move your mouse over them.....

STEP 1 :

The first step is to download ExpressPCB from the internet and install it on your PC (it will run with Windows XP through Windows 7). It is a free download and is available from http://www.expresspcb.com (http://www.expresspcb.com) . When you go to install it, it will install two applications : ExpressPCB (which is used for creating PCB layouts) and ExpressSCH (which is used for drawing electronic schematics) and place shortcuts on the Windows desktop. Of course the reason that it is free is that they anticipate that you will do an 'on-line' order for your PCBs through them and they will make their money that way.

I have no intention of describing how to use the applications here because there are help files included even if not totally clear the first time you look at them. There are also a number of components included in the supplied libraries but I have found that I have had to create extra 'custom' components that are more oriented towards my specific areas of interest, particularly with some of the newer Surface Mount Devices (SMD) layouts.
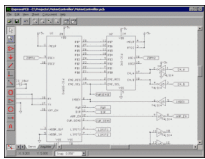

The people at ExpressPCB recommend that you begin your project by drawing a schematic. While not required, it will save you time when designing your PCB. Drawing a schematic with the ExpressSCH program is supposedly as easy as placing the components on the page and wiring the pins together but that is only true after you have developed experience with the program . The schematic can then be linked to your PCB file, so that the PCB knows what needs to be connected together - but - I find that I don't necessarily want to draw a schematic electronically if one already exists on paper - eg in a book, magazine or even a freehand scribble on scrap paper.
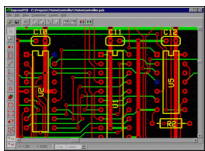
The people at ExpressPCB : Designing 2 or 4 layer boards using the ExpressPCB program is very simple. Start by inserting the component footprints, then drag them into position. Next, connect the pins by drawing the traces. If you link your schematic file to the PCB, the ExpressPCB program will highlight the pins that should be wired together in blue.

As stated a few paragraphs above, you WILL find that you need to create custom components for either/both application simply because the libraries are not expansive. It isn't difficult and once you get the hang of it then it does seem simplistic. You can also modify the current library components and save them with your alterations as 'custom' versions with new names.The PCB layout illustrated above is beyond the abilities of the simple home hobbyist BUT if you only require one copper layer (single sided) or maybe two layer (double sided) and are prepared to use wire links to connect parts of the PCB then you can do it at home - and at a reasonable cost.
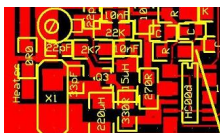
Actual size of this piece of the PCB is just 30mm x 20mm high (so around a square inch : 25x25mm)
RED = TOP of PCB
*YELLOW = Component overlay layer*

This is a small corner of a new PCB layout that I have just finished and it uses primarily Surface Mount Devices on the upper side of a single sided PCB. In actual practice, the PCB material is double sided and the second side is left as a copper (earth) backplane but all components are mounted on just one side - on top. There is no need to use [plated-through] "vias" (connections through from the tracks on one side of the PCB to tracks on the other) if the project is not too complex and wire jumpers are used. One trick that I have used in this particular layout is the use of SMD 1206 size 0R0 resistors (ie zero resistance links) to jump across a PCB track or two instead of needing jumper wires.

The standard size SMD resistors that I use are 1206, most of the capacitors up to 100nF (and some up to 1uF) are 0805 size, and RF chokes (220uH above) are 1210 size. The component X1 is a crystal and the circle above it is a trimmer capacitor - both of which are custom components. The transistor Q3 is an SOT-23 case style and the component marked HC00d is a 14 pin SO14 ' 7400 High speed CMOS series ' quad nand gate IC. The size of the slice of the PCB shown above is just 30mm wide x 20mm high in practice and the narrowest track width I used was 0.25mm (the vertical lines/traces between the HC00d pins are that width). You will note that there is mostly red (copper) and little black (insulating sections) in the above image because most of my circuits are used at radio frequencies where shielding and low interconnection inductances are mandatory.

I use widths down to 0.25mm for fine track work and 0.38mm for general layouts. If you follow the instructions carefully when processing the PCBs then you will have no trouble re-producing PCBs with the finer 0.25mm widths. Text labels on the parts overlay are normally 0.97mm.

The whole process is a lot easier if you are already following the PCB layout in a magazine. If you can extract a high quality full-size (ie 1:1) scale of the PCB then use that. Alternatively use ExpressPCB to duplicate the layout printed in the article - after all they have done the hard work in laying out the components for you and a 1:1 artwork will be the result if you use properly sized components (eg from the supplied libraries).

These are my current (at Feb 2010) custom component libraries for PCB (/~vk4adc/web/images/UserFiles/File/pcbs/pcbcustom.zip)and Schematic (/~vk4adc/web/images/UserFiles/File/pcbs/schcustom.zip) (about 16KB each, right-click & Save As). You will need to extract them into typically *C:\...\My Documents\ExpressPCB\PCBComponents_Custom* and *C:\...\My Documents\ExpressPCB\SchComponents_Custom* respectively (these folders are created automatically when you install the Express software) to be able to use any of them in your projects.
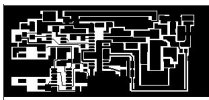
Top layer only is laid out using ExpressPCB
Bottom layer for this project is plain copper.
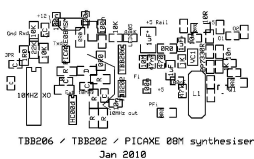
This is the PCB overlay to suit the above layout.

STEP 2 :

Let's now assume that you have the layout on the computer and want to produce the artwork. Note that we require a "Positive" - ie where PCB tracks are required we have a black mark/outline. Where fibreglass insulating segments are required (eg between tracks), the artwork is "clear". ExpressPCB will only print a positive artwork so that makes it a bit easier.

I have found that the best quality artwork actually comes off a good inkjet printer using medium-good quality transparency media. I have tried laser printers at 600dpi and laser/copier transparency materials and the resulting prints are not really dark/dense enough. I currently use Celcast a pack marked IJ21-50 (50 transparencies) and an A4 Epson inkjet with the media setting at "Ink Jet transparencies" and print quality as 'Best Photo'. Note that there are no printer settings within ExpressPCB so you must set the default values for the inkjet to these BEFORE you start the ExpressPCB software. Always print the artwork to normal plain paper first and double check the print quality and for any layout or sizing errors BEFORE you put any transparency media in the paper feed of the printer !!!!! Paper is cheap, transparencies are expensive in comparison.

I always turn off the yellow overlay view at some stage when I think I am finished so that I can see any obvious errors on the top layer - like tracks that aren't actually connected - but this fact is usually obscured by the yellow lines and text of the component overlay.

My SMD layouts are usually 'on top' so therefore I almost always print the TOP layer then print the Overlay on plain paper expanded to fill the page so that I have the parts layout to work from when installing the components.

The visual test when you hold the final positive up to light is that the blacks are reasonably black and the clear is actually clear.

STEP 3 :

You will need to have purchased positive resist pre-sensitised PCB. Probably the most-readily available source Australia-wide (or world-wide) is the that from RS Components (http://australia.rs-online.com/web/home.html). There used to be a number of hobby stores like Jaycar and Dick Smith Electronics that had positive resist board readily available but the move away from electronics home-construction probably sounded the death knell for the demand.

On the **RS Components (http://australia.rs-online.com/web/home.html)** web site, simply enter " Fotomechanix " in their search box. The results will show around 20 products starting at sizes around 100x160mm in single sided up to 305x457mm in double sided. My preference is for the product coded 397-0053 (http://australia.rs-online.com/web/search/searchBrowseAction.html?method=searchProducts&searchTerm=397-0053) double sided FR4 positive resist PCB material, 203mm x114mm - about $18.50 + GST = $AUD20.35 (early 2010 pricing). Simple projects can use 397-0047 (http://australia.rs-online.com/web/search/searchBrowseAction.html?method=getProduct&R=3970047) single sided FR4 positive resist PCB material, 203mm x114mm at $17.30 + GST. The good thing about RS Components is that if you are in a capital city with a Trade Counter, you can just buy these over the counter rather than having to place a web order.

This PCB material is covered on the sensitised side(/s) with an adhesive black plastic layer so that makes it insensitive to light until the layer is stripped off exposing a greeny coloured copper underneath, and that happens just before you expose it to light through your artwork.

The plastic layer also make it easy to cut to size because you can simply use a standard hacksaw with a fine (32TPI) blade (or maybe even a jigsaw with a fine blade) to trim it and still handle it in full light conditions. Simply draw a pencil line using a 90 degree square on the plastic and it will leave enough of a mark to cut along. Just make sure you don't actually flex the PCB material and cause cracks in the resist layer/s.

Make sure that the material you cut is larger than your artwork, preferably about 10-12mm longer and wider (so 5-6mm is available on each side). Once you get the hang of it then these overlaps can be reduced but it does make it easier for the beginner if you make the PCB larger and later trim it to size.

***** Note that you can produce true double-sided PCBs but you have to take a lot of care with side-to-side registration and usually expose both sides similtaneously.
This page will not deal with the methodology but some other web sites do. *****

WARNING : FROM THIS STAGE UNTIL AFTER ETCHING (Step 7), DO NOT TOUCH THE PHOTOSENSISTIVE SIDE OF THE PCB - HANDLE ONLY BY THE SIDES.

STEP 4 :

The next procedure is to expose the sensitised PCB to light through the artwork.
I use an old contact print frame but if your need is to create very occasional PCBs then a suitably-sized clean piece of glass will suffice. { an A4 or A5 picture frame is a good source of glass this this purpose. } The contact print frame is simply a rectangular piece of pine to which is hinged a rectangular pine frame for the piece of glass. The glass is simply glued around the edges to the rectangular frame with a contact adhesive like KwikGrip or GelGrip. A nearly-the-same-size piece of thin foam rubber glued to the backing timber provides a "sandwich effect" and will stop the PCB material from moving. To use a contact print frame, the PCB material is placed on the foam rubber, the artwork centred on the PCB and then the upper frame is closed down by virtue of the hinges.

The step I didn't disclose in the last paragraph is that the plastic layer must be removed off the PCB in subdued light or under yellow light. Do not do it in bright light or the board will already be exposed directly rather than through the artwork. I tend to do the plastic removal in a darkened room (ie. lights off) away from any windows, immediately position the artwork on the top of the 'green' PCB, close the print frame and IMMEDIATELY cover the whole lot with a thick cardboard (I use a manilla folder - it doesn't have to be black).

I use a typical exposure of 20 seconds to direct sunlight to expose the board. I simply take the cardboard (/manilla) cover off the print frame for the exposure period while making sure that the assembly is perpendicular to the sun itself. After the exposure period, you just pop the cardboard back over it and take it back inside. If you want to avoid using the sun, you can expose the PCB to suitable UV light sources (around 360nM) for longer times but the actual times will be determined by the actual bulb used and the distance the light is away from the PCB during the exposure. The sunlight option is easier except on rainy days or if you can only do it at night.

The 20 seconds has proven accurate for direct sunlight here in Brisbane Australia but other areas of the world may require different timings.

Note that even though I generally use double sided PCB material, only one side is actually exposed and etched for most of my projects. The second side remains as a full copper earth plane and the adhesive photo-blanking material is left on it right up to step 8.

STEP 5 :

The developing of the exposed PCB is next in the process and again you need to do this under subdued light or yellow light.
I have had great success developing in a weak caustic soda solution with a development time of about 10 seconds. You don't need to buy the RS universal developer product 680-949 (http://australia.rs-online.com/web/search/searchBrowseAction.html?method=getProduct&R=0690849) unless you want to..

The caustic soda (Sodium Hydroxide / NaOH) that I use is granular and is commonly used for clearing blocked drains in the kitchen or bathroom, available from the supermarket or hardware. The product is sold here under the brand name Mechanix and is supposedly 98% NaOH (see suppliers web page for info (http://www.pascoes.com.au/faq.php?group_id=14)) but of course any high quality NaOH source could be used. The important thing is that you must use cold water and slowly add the caustic soda to it - NEVER add water to caustic. The resulting solution will grow warm as the caustic dissolves.

From the RS product info : Developer concentration: 25g/l of 690-849 (http://australia.rs-online.com/web/search/searchBrowseAction.html?method=getProduct&R=3970097) or 6-10g/l of Sodium or Potassium Hydroxide. Conduct trials if in doubt.

The quantity I use is about 1 teaspoon of caustic granules to 1 litre of water. The right consistency is when if you wet your fingers with the solution, rubbing the fingers together will feel soapy but not unduly "bite-y" (/ burning). This solution is made up in a rectangular plastic container about 30-50mm deep and of other dimensions to suit your PCB size. In some cases, I pour half the solution off before using. Do NOT make the solution too strong or else it will rip the entire photo layer off the board. Just remember that you can always add a few extra granules to the mixture... after first removing the PCB !

Still under subdued light, uncover the exposed PCB, remove from the print frame and quickly drop it into the developer. Almost instantly you will see swirls of purple or green and sometimes almost black emanating from the surface of the PCB as the exposed-to-light sections of the photosensitive coating are reacted away. As soon as there is no more purple / green coming off the board and the board looks 'finished', wash it under cool running tap water and inspect the board carefully for any section not completely developed. If it is ok then you are finished this step otherwise continue developing.

The weak caustic solution can simply be poured down the sink after use and washed through with running water. After all, it IS sold for clearing kitchen drains !
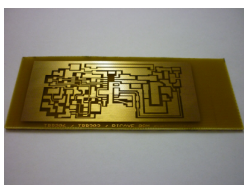
After developing

STEP 6 :

The etching and the remainder of the processing is done under normal light conditions.

The etchant is ferric chloride (FeCl) and is available from RS (216-2699 (http://australia.rs-online.com/web/search/searchBrowseAction.html?method=getProduct&R=2162699) but it is 1Kg at $50+), Jaycar, Altronics (etc..) have it in liquid form at a lower cost ..
Do NOT use Ammonium Persulphate as an etchant : you have been warned ! **See notes at top or bottom of this web page.**

Before you etch, cast a second look at the developed PCB material for little dust spots or scratches in the resist layer. The dust spots can be removed from insulated areas using a sharp blade and scratches and marks on the resist repaired by the use of a Dalo pc-resist pen. Please allow the resist pen ink to dry properly before you start the etching process.

This stage uses two rectangular plastic containers, one that fits comfortably within the other. The smaller one has the ferric chloride solution with a depth of about 4-6mm and the larger one has boiling water from a kettle poured into it. This arrangement allows the ferric solution to be heated and made more efficient as an etchant without diluting it. The smaller container basically "floats" in the hot water bath.

The developed PCB is simply placed into the etchant and the inner container "rocked" in the hot water bath, a process that I do in the kitchen sink (just as well my wife doesn't know about this !) .
**This movement produces a swirling motion of the etchant over the board and you will see parts of the board exposed from time to time. The exposed copper quickly becomes a dull copper finish and as the copper is eaten away by the ferric, the copper colour changes to a grey and finally to that of the fibreglass itself where there are no tracks. When the board is completely etched, the pattern should match exactly that of the positive artwork.**

A word of caution - do not leave the PCB in the etchant for 1/2 hour while you wander off to have a cup of coffee. Very fine tracks can be side-eaten under the resist layer so it is important to remove the board as soon as it is completely etched and then wash it under cool running tap water..

Tip : Ferric chloride stains everything browny/yellow - fingers, sinks, table tops... you need to wash it off your fingers or the spills into the sink quickly with running water ! Dilute, dilute, dilute are the 3 key words.

Do NOT pour out the ferric chloride into the drain, just put it back in the plastic or glass bottle, plastic lid a necessity, for next time.....

For my PCBs etched in the heated pre-used (& therefore old) ferric chloride solution, the etch time is only about 8-10 minutes.
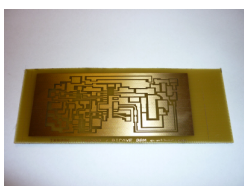
After etching


STEP 7 :

Removing the resist layer .

The RS info sheet on the Fotoboard2 material states : "The unexposed resist can be left on the board to act as a protecting film if required. If a solder joint is to be made, it can be done without removing the resist."
This is an issue for me as I usually use Isopropyl Alcohol to clean off any residual flux after soldering, and as you note above, this also softens & cleans off the green resist layer !!!!

The positive resist green coating washes off with Isopropyl Alcohol & paper towel - don't buy the RS stripper/cleanoff product 690-855 (http://australia.rs-online.com/web/search/searchBrowseAction.html?method=getProduct&R=0690855) unless you want to.
I usually just pour a small pool of the Isopropyl Alcohol (from Jaycar, Altronics etc) onto the PCB top surface and then use a paper towel to wipe off the coating.

You can use a scourer to remove the layer but that is hard work - and you actually remove some of the copper layer when you undertake this action.
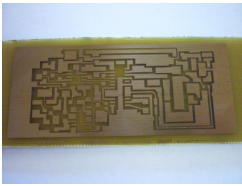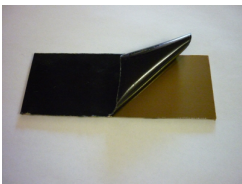

After cleaning off resist from main side

STEP 8 :

For single sided material, jump this step.

If you used the double sided material, remove the adhesive protective layer now and wash off the second resist layer with Isopropyl Alcohol.

Removing protective cover from side 2

STEP 9 :

Trim the PCB to the size of the artwork.

Use a hacksaw, file etc to cut to the required size BUT make sure that you use "soft jaws" if you place the PCB in a vice so that you do not damage the PCB tracks.

The best method is to use a metal guillotine but not too many of us have one of these at home.

Note : "soft jaws" are typically either L-shaped 1mm+ thick plastic or soft wood (eg pine) pieces that are put into the jaws and the 'job' placed between them. Their job is to protect the object's surface from damage.

STEP 10 :

Spray a protective coating of clear lacquer on the exposed copper layer(/s). This will prevent tarnishing of the copper and give it a longer lifespan.

I generally use a PCB coating spray, eg Electrolube Clear Protective Lacquer, which can be readily soldered through.

If I am in a hurry then I sometimes run a hairdryer or a heatgun (on low) quickly over the sprayed side to make the lacquer go off immediately.
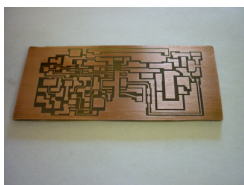
PCB after trimming & spraying with clear lacquer

That's all. Once the PCB's coating is dry you can start drilling / soldering to make your project complete.
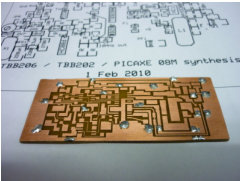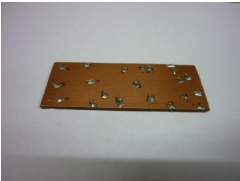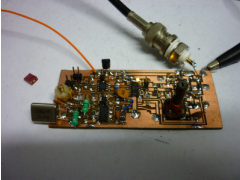
It usually takes me about 20 to 30 minutes to cut, expose, develop, etch, trim and lacquer a PCB. Of course the PCB layout takes a reasonable amount of time but once you have the artwork, producing extra PCBs with a given design is just so simple.

The RS web pages also list a Positive Resist in a spray can 198-9651 (http://australia.rs-online.com/web/search/searchBrowseAction.html?method=getProduct&R=1989651) that you can to spray onto clean/prepared standard PCB to make your own sensitised board.

I have plenty of standard PCB material in both double and single sided forms and have tried some of the spray, baked it, and done as per the instructions but it just seems to wash completely off during the developing process. I have even tried multi-coating the boards - with the same result.
I really haven't had much success with this product but maybe you will as it does reduce the cost by making your own......

These are further images of the above printed circuit board project :

Solder-through "vias" to connect the earth lands on the top to the bottom ground plane	Bottom view with "vias" soldered in place
The PCB board up and running.	Yellow LED at centre top lights under certain conditions

----

Finally, the more you make up PCB layouts and then go ahead to produce the physical PCBs then the easier & faster the process gets - and the fewer the number of production failures .
i.e. 'Practice makes perfect'.

Nov 2010 :

In the months since I first created this web page, I have produced many PCBs for my projects. I have had an occasional failure in the due processes but those can be attributed to...

- Incorrect exposure (eg clouded sun) - used clear sky sunlight only
- Incorrect exposure times (eg trying to correct for clouded sun..) - about 20 - 30 seconds of clear sun is enough.
- Incorrect developer strength - too weak and too strong - see the strength specified in section 5.
- Improper board layout - tracks closer than 0.25mm to other tracks or PCB lands.

Many of my layouts use 0.25mm tracks and I have successfully and consistently run this width track between the pads for an SOIC IC. Yes, you have to be careful when soldering the components on but the PCB itself has been perfect.

My most recent PCB-based project utilises 3 x 8 pin SOIC, plus 1 x 14 pin SOIC, plus multiple resistors and capacitors, all on a 42mm x 17mm single sided PCB. Yes, 0.25mm tracks in some places, no wire jumpers - but a few 0R0 resistors placed to bypass the need for them.

Of late, I have tended to leave the protective photo resist layer on the board, rather than wash it off with the isopropyl alcohol. It is easy to solder through but take time and care to do your joints properly.

Most enquiries received in relation to this web page are about the use of ammonium persulphate in this process. I can't find the note on the RS website at the moment but it is there somewhere. As a result, I have added the notation at the top of the page itself and repeated here :

Note that ammonium persulphate **can** be used to etch the copper laminate on PCB material created using most other methods (eg iron-on...) --- **however** --- RS Components warns against using this chemical on the Fotomechanix / Fotoboard2 series of products, the ones specified in step 3. They specify ferric chloride as the etchant - take note of this if you intend using this PCB materials range from RS.