Henry Carl Ott N2RVQ carlott@rcn.com TTS REV 1.1 30-SEP-2001

Introduction:

Here are some notes on my experiences with Toner Transfer System (TTS) material in the home production of Printed Circuit Boards (PCBs).

Ever since I've been involved in electronic design I wanted a mechanism to allow me to easily and economically produce prototype quantities of printed Circuit Boards (PCBs).

This was not a lone pursuit. While not exactly the Holy Grail of electronics, anybody who has ever done any design of any complexity understands the attraction of a pcb to aid in construction. Prototyping with perf board is fine until you want to mount a part with a non .1 inch pin spacing,(a PC mount DB-25 for example). And now with surface mount parts actually becoming standard, a PCB becomes a necessity for many designs.

There are other advantages, but for me the fact that I can have an idea in the morning and then have a completed populated PCB by the afternoon is the main attraction. I used to only bother to make PCBs of a certain complexity, or to test a design before it went out to a PCB fabrication house, but now I'll make a PCB for a design that just needs a single resister and an LED.

Other methods exist, but I feel that there are only a few options for prototyping boards in-house. Some of these are theoretical or maybe not practical, but people always talk about them.

"By Hand" methods:

1. Mechanically.
Lay out your board and use a x-acto blade and or Dremel tool to remove excess copper.

Can't get any simpler then that. It works, and you don't use any etchant!. But, there is lot of time involved. I've done some very simple RF design this way, however it really is not practical for design of any complexity.

2. Resist pen.
Draw directly on the copper with a resist pen, then etch it.

This will work and is pretty cheap, but results are sloppy and you'll go nuts if you have to make more then one board. Again this is not suitable for complex boards.

Computer assisted manufacturing (CAM).

These methods assume that the PCB design was done on a computer.

1. Mechanical copper removal.
The output of your design is processed by software, which then drives a custom xy table with a high-speed cutting head that removes copper to electrically isolate signal paths. If you give it more time it will remove all the excess copper to make a more conventional looking board.

No etching solution! And I believe the table can also be used to drill your board, and do engraving. Nice! These are great systems but are very expensive. I can't justify the cost as an individual, but for a business this is a very viable solution.

2. Directly plotting to copper.
Computer output directly drives a flatbed XY plotter that draws with waterproof ink directly on the blank copper clad board. You wait for the ink to dry them plop the board into the etching tank.

Sounds simple enough, but when we tried it we had lots of problems. Problems with the ink, problems keeping the board flat. We gave up because the yields were terrible. You could probably make the system work with some sort a vacuum system to hold the board flat and a pressurized ink delivery system.

3. Photographically applied resist.

Computer output is printed on a transparency sheet. Pre-sensitized copper clad is exposed to UV light through the printed artwork, board is developed via a simple process, then etched.

Works great once you determine a couple of variables such as exposure time. This is basically the process the large PCB fabrication houses use. Once you have the artwork it's easy to make multiple copies of the board. This is a very viable option for in house prototyping. It's slightly complicated by the necessity of board exposure and developing, but my only real complaint is the expense of the pre-sensitized boards. Trying to apply the photographic resist laminate yourself to your own copper clad is difficult and or expensive.

4. Toner Transfer Systems.

Computer generated artwork is printed by a toner based printer onto special paper. Heat and pressure is used to transfer the toner to copper clad, which is the put into etching tank.

And here we come to the main topic of this document.

I've been using toner transfer systems to fabricate PCBs for the last 8 years with good to great results. I'm basically going to try to document my experiences so other people can benefit from my experimentation.

NOTE: I use this system primarily to make single sided boards. You can do double-sided boards, but its a bit more work aligning everything. Also plated through holes (PTH) are incredibly difficult to do at home. You can use very small eyelets but it again it requires much effort. If you want additional info on double-sided fabrication I suggest you look at the very informative DynaArt site.

NOTE: Key concept:

The reason the system works is the fact that, printer toner is just basically just powdered plastic. Plastic that is water proof and can be melted and re-melted. This is why only toner based printers/copiers and not ink jet printers are suitable for this process.

The process is basically:

1. Generate the artwork on your computer.
2. Print out the artwork via a toner-based printer onto Toner Transfer Paper.
3. Use heat and pressure to apply the toner transfer paper to blank copper clad.
4. Soak board and transfer paper in warm water to remove paper and leave toner.
5. Inspect copper clad and touch up if necessary with a resist pen.
6. Drop board into etching solution until all unwanted copper is removed.
7. Remove toner, drill board and populate.

I will try to break these down into specifics:

First: Generate the artwork on your computer.

Use any software you like. This can be a simple paint program, or a dedicated PCB CAD package. There are free or limited demo versions of professional PCB design packages that are perfectly suitable for small designs. Check the resource section for some examples.

It is helpful if the software has the capacity to print mirrored output. This is necessary because the transfer process effectively mirrors the artwork and we have to pre-mirror the output to get the results we want.

When you do your design be sure to have some text on the board (even if the text is just 'solder side') that be used to help keep track of the reversals and insure that the final product is correct.

Second. Output the artwork onto Toner Transfer Paper.

After your design is done, print it onto the transfer paper.

NOTE: There are multiple sources for the actual transfer paper and we will discuss that elsewhere.

NOTE: Output normally must be mirrored!

NOTE: The printer must be a toner based printer. This could be a laser printer or an Okidata led printer. If you only have access to a ink jet printer you can print it out normally, and then use a toner based copier (xerox) to transfer the design to the transfer paper.

I've used HP Laserjet series II, IIP, and III with good results. In theory higher resolutions printers should give superior results, but I have no experience with them.

NOTE: It is also usually necessary to increase the print contrast all the way up to get as much toner as possible onto the paper.

NOTE: Some printers have problems printing calibrated output. If your printer is old you may find that either the x or y axis does not print accurately. If it's only off a small amount you won't really notice it except on large (multi pin) connectors. Larger errors will cause problems, you can try rotating the output to move the problem dimension into a less critical area. Some cad packages allow you to calibrate the output and enter in a error correction factor. If none of these work you may need to find another printer or figure out a way to post process the output to adjust for the error.

Since transfer paper is more expensive then regular paper, you may want to use certain techniques to not waste any. Unless your design is 8.5 x 11 inches there is no reason to use an entire sheet of transfer paper for a small board. We can use a carrier sheet to actually run a smaller piece of transfer paper through the printer.

Have your cad package print your design centered on a regular piece of paper. Then cut out a piece of transfer paper slightly larger then the actual design (a half inch border is sufficient). Place the smaller piece of transfer paper onto the first "carrier" sheet using the printed design output for alignment. Be sure to place the transfer paper correct side up! Using a small piece of masking tape (or other tape that won't melt in the printer) secure the smaller piece of paper only on the edge that will first enter the manual feed of the printer.

Now reprint your design and feed the combined carrier sheet and transfer paper through the printer.

Recover the printed design, discard the carrier sheet and if necessary trim any excess paper to leave a small border around the artwork.

Third. Use heat and pressure to transfer the toner to copper clad.

. I use single copper clad almost exclusively for prototyping. I pick it up as cheap surplus in varying thickness. I've also had good luck asking for odd sizes and scraps from the PCB Fabrication houses that I use for larger production runs.

Prepare your blank copper board material. I just clean it up to bright finish with fine steel wool. Scotchbright or something similar also works fine

NOTE: Don't use steel wool around computer equipment. After cleaning the board you get little pieces of steel wool all over the place. You don't want these potential shorts around electronics.

NOTE: Be sure not to contaminate the surface after cleaning! Don't touch the surface with your fingers and only handle the board by the edges.

NOTE: Okay here is the first really critical piece of advice. Despite what people say, do NOT use a standard clothes iron to re-melt and transfer the toner to the copper clad. There is no way to get consistent temperature and pressure.

What you really want is a heated roller laminator. They make these to laminate business cards into luggage tags, make identification badges and restaurant menus. You can find them at stationary and office supply stores in varying sizes. I usually try to pick them up cheap used ones at hamfests because they seem to be a little over priced for what they are if you buy the new. Note: If you can, find one that has a temperature adjustment. I've been using a badge laminator that I picked up cheap with great success. It's limited to boards a maximum of five inches across, but that's fine for the bulk of my designs.
As an added bonus I can now make my own luggage tags, and laminate all my friends Ham tickets.

NOTE: You will have to adjust the temperature to get good results. Toner compositions vary by manufacturer so you may have to do a little initial experimentation to get it right. I just cranked the temp. way up on mine and it seems to work fine. Sorry I can't give you an exact temperature.

NOTE: Depending on the exact laminator design you may have problems with really thick PCB material.

NOTE: I run the board through the laminator twice just to insure results. A single pass may be all that's necessary.

NOTE: DynaArt sells a modified laminator that's optimized for PCB production. It's a very nice piece of equipment, you'll just have to decide if you want to pay the price.

Fourth: Soak board and transfer paper in water to remove paper.

After the board comes out of the laminator, let it cool to room temperature.
Then place the board in warm water. After a couple of minutes the paper should just float off. If the paper does not come off on it's own it's okay to gently peel it off after a few minutes.

If you are using the EPSON paper you must then gently clean excess backing material off the board by gently rubbing with a soft bristle brush.

Fifth. Inspect copper clad and touch up if necessary with a resist pen.

After the board comes out of the water let it air dry.
Carefully inspect the board. If necessary use a fine point Sharpie permanent marker to touch up any breaks. There should not normally be any.

NOTE: Larger unbroken areas of toner may not be solidly filled in. If necessary go over these areas with the marker.

Sixth. Put board into etching solution until unwanted copper is removed.

NOTE: For safety purposes always wear gloves and protective eye wear when dealing with caustic chemicals.

I use two etching methods.

For just one or two small boards I use ferric chloride in a plastic tray that is attached to laboratory agitator. Ferric chloride works well at room temperature, but will stain anything it touches.

If I'm etching larger boards or if I know I'll be busy making boards for the next couple of weeks I mix up a gallon of sodium persulfate. Sodium persulfate comes as a dry powder. You mix a pound of it to a gallon of water to make a working solution. I use the sodium persulfate in a wave etching tank that I purchased from DynaArt. Sodium Persulfate is easier to work with then ferric chloride, but it must be heated to achieve good results. It also has a limited shelf life once it's been added to water.

The DynaArt wave tank works well. I had some initial problems with the magnetic impellers in the pumps deteriorating. This was resolved when it was discovered that it was not a good idea to store etchant in the tank for extended periods of time. If you are not going to be using the tank for more then a couple of days then drain the etchant into a storage container (if it's not spent it can be reused) and then add some clean water to the tank. I prefer to store the tank wet, but if don't have the room I'd guess it's okay to store it dry.

NOTE: DynaArt has a newer model wave etching tank. It incorporates some design changes that make adding and draining etchant far easier. I don't own the newer model, but I wish I did. I may make some of the design modifications to my older model.

NOTE: Always dispose of the used etching solution safely and in accordance with local regulations.

Carefully watch the board as it etches. When all of the unwanted copper is gone take it out of the etching solution immediately. If you leave the board in too long, you will get under cutting as the etchant dissolves the copper from under the protective toner. Clean the board thoroughly under running water and you are just about done.

Seventh. Remove toner, drill board and populate.

At this point the board is almost complete.

You can trim the board to exact size. But you want to leave the toner on the board until you are ready to assemble it. The toner protects the copper from oxidation.

When you are ready to assemble, remove the toner with steel wool or acetone.
Drill the board, and populate.

I use a small jeweler's drill press to drill the boards with some surplus used carbide drill bits.

NOTE: It's impossible to drill the board with these small bits by hand. You need a good small drill press or a Dremel tool in a drill press adapter.

NOTE: Be sure to wear protective eyewear when drilling The carbide drill bits break all the time and go flying everywhere.

Notes on different toner transfer paper:

I've used four different transfer papers. Three are sold specifically for making PCBs and one is a pure hack. Here are some notes on each of them.

Techniks sells two different type of transfer paper.
1. Press-N-Peel Blue, which is a dry process. After the heat process you just let the board cool and peel off the sheeting. In theory, the toner stays behind. It's not really paper, but a plastic sheet coated with a release compound.

This was the first transfer system that I had any luck with. It works okay, but seems a little more sensitive to process variations. I've had various problems with toner not sticking properly to the copper or smearing. I've pretty much switched over to the 'wet' process.

NOTE: I've had problems with the Blue material smearing/bluring traces. If too much pressure is applied the tones melts and reflows. The plastic backing squeezes the toner into a thin paste and ruins the results. I think that paper backing work better because it provides a buffer to prevent this smearing.

2. Press-N-Peel wet. Pretty standard toner transfer paper. Print, Laminate, Soak, and then etch. Works well, and is cheaper then the equivalent DynaArt product.

DynaArt sells a high quality Transfer paper. It works just about the same as the Techniks product, but costs twice as much.

DynaArt also sells all sorts of supplies and tool to make home PCB fabrication easier. You should definitely peruse their web site. Highly recommended.

3. Epson Photo Quality Glossy Paper. Part No.S041124
Okay, this is a definite hack. After following up on a tip posted in a usenet group I purchased some of this Epson Paper. It works as well as the other papers but at half the price (about $12 USD for twenty sheets). It may also be more accessible to some people. It works the same as the other papers described, but there is an additional step. After the board is soaked in warm water and the paper releases there is still some

backer material left on the board. It is a particular problem in areas of fine detail. What you do is let the board soak for a few more minutes, then remove the excess material by gently rubbing the board with your fingers or a soft bristle brush. The toner is really attached to the copper, so you'd have to rub very hard to dislodge the protective toner.

NOTE: It has been suggested that a ½ inch border of toner around the actual design may improve results.

Advantages:.	It's cheap.
Disadvantages:	A little extra effort to remove the backer material.
	Epson did not design this paper for this application. They may in the future discontinue the product, or change the specification to make it unsuitable for this process.

Unresolved Issues.

At the moment I just have one problem with the system. Laser printers really don't want to print large areas of black. Hold the output of a test sheet up to the light and you will see that the black is not black at all but actually gray. The toner transfer system faithfully transfers this non-solid toner to the copper, and when you etch you get some pitting.. On really humid days this problem seems to get even worse. I get around this by not having solid ground planes, instead using a hatch pattern for the plane. If anybody else has experienced this problem and has a solution, I'd love to hear about it.

Resources:

Suppliers of toner transfer paper and other fabrication aids.

DynaArt
http://www.dynaart.com

Techniks
http://www.techniks.com/

Suppliers of free or usable demo versions of PCB layout software.

Cadint
http://www.cadint.com/

Cadsoft
http://www.cadsoft.de/freeware.htm

Protell EasyTrax
http://www.protel.com/etech/freeware_home.html

Bob Arnold sent in some feedback and suggested Vutrax as a very solid design package:
http://www.vutrax.co.uk/

Revisions:

1.0 Initial release
1.1 Added Vutrax recommendation. Minor edits