Home PCB Production – v0.0.2

Sasha Zbrozek

Introduction:

The current purpose of this document is to aid in the manufacture of printed circuit boards with limited materials and equipment. It covers supplies, procedures, and some finer details.

Designing for a homemade process:

Homemade circuit boards done right can handle traces as thin as 5 mil, but it's advisable to use thicker traces whenever possible. Many things can happen to foul a board, and thicker traces are more resilient to error and damage.

Make 45° bends wherever possible. The larger radius of curvature turn is less likely to peel when removing the paper during the artwork transfer process. It also just looks nice.

Switch to oval or rounded rectangle parts pads wherever possible. The rounded edges aren't prone to peeling nearly as much as rectangular pads. The corners start to come up and before you know it the whole pad is down the drain.

Make really big vias and holes. Homemade vias are inherently huge. I make via holes 30 mil and pads 60 mil. I use a drill press to drill them, and very small gauge wire to make via connections. I edit the footprints of DIP parts to have wider pads to accommodate my drilling technology.

Turn on pilot holes when making Gerber files. They're invaluable.

Of course these are just guidelines and not requirements. Even in my own examples here in this guide, I did not follow all of these tips.

Choosing the paper:

Paper is the transfer medium from the printer to the circuit board. A number of properties influence the quality of the process. The paper should be multilayered. The top layer should be a polymer (plastic-like) coating engineered to capture inkjet spray without blurring. The intermediate and backing layers should be paper based so that the sheet will dissolve and tear off easily. There are some purely polymer sheets out there. Avoid them at all costs. Their plastic nature makes them difficult to tear, impossible to rub off, and they warp and melt under the iron. They're terrible.

Glossy photographic paper seems to be the best choice for this. The paper's top layer should be as smooth as possible to prevent pits and other irregularities in the transfer. The polymer coat should also be able to withstand a few minutes at the high temperatures required to melt the toner without excessive curling. For United States and some international consumers, Staples Basic Glossy works very well and is reasonably priced.

Coated paper is adequate, but the clay particles tend to cause flaking of the transferred toner when removing the paper. Some types of magazine paper work well, but the coatings tend to gum up laser printers. Plain paper is abysmal, it's very rough and doesn't easily let go of toner. Plain paper is not even an option, it's futile.

Choosing the printer:
Not just any printer will do! Always use a laser printer that has chemically precipitated toner (CPT) if that technology is available. Attrited toner will work, but it should not be used in any ambitious projects. The reasons are numerous. CPT under a microscope looks like a big pile of marbles – all of the particles are spherical and uniformly sized. Attrited toner looks like an asteroid field – all of the particles are of irregular shapes, are often pitted, and have a strange size distribution. CPT toners are narrowly centered on a 5µm particle size; attrited toners are widely centered on a 15µm particle size.

But how do these factors impact circuit board production? The smaller particles allow finer surface features with less blurring. The uniform particle size means fewer gaps in fills, better adhesion, regular melting, and a number of other bonuses. The spherical shape means better packing and therefore fewer gaps and a stronger final etch resist.

Picking specific models of printers is outside the scope of this document, but to date only a few manufacturers use CPT. Included are Hewlett Packard, Canon, Samsung, Konica Minolta, and possibly Kyocera Mita. Lexmark does *not* use CPT, so avoid them. Be sure to check the specifications and marketing literature for your printer and cartridge to determine what type of technology it uses.

If buying a new printer, also consider the paper path. A simpler D-shaped paper path will accommodate heavier papers with less paper bending. I use a Konica Minolta 1350w. It was $50 at Office Depot and comes with a toner cartridge. I use it exclusively for circuit boards, and so I'll never need to buy a new cartridge.

Preparing the art:
When possible, produce 600 DPI artwork. That's the usual internal graphics engine resolution of laser printers and results in a minimum of rescaling and interpolation. Some printers and printer drivers will also accept and work with 1200dpi natively, but many just rescale to 600 dpi. A "1200dpi" printer may be able to print with that resolution, but they may not be able to accept data at that resolution due to the larger memory and CPU requirements of handling the image data. That doesn't prevent marketing departments from latching on to the biggest available dpi number and pasting it all over their product literature.

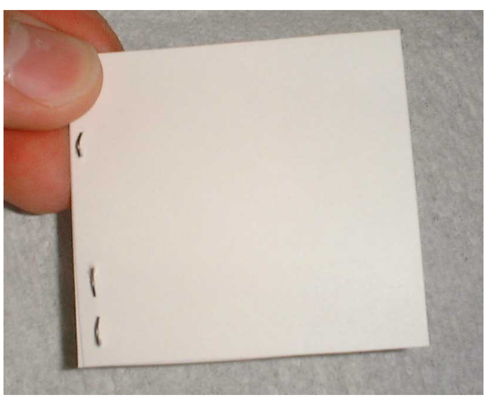
Spatial orientation is very important when preparing the artwork for printing. Consider the page flips required to position the toner side of the printed paper on the appropriate copper sides. Usually the top side copper must be mirror imaged.

For software, I use The GIMP or Photoshop, depending on where I'm working. In either case, it's easiest to create a blank file the size of your paper and put the artwork in that blank file. Rotate and mirror the two halves until they look like they could be folded together off of the page to make your circuit board. I leave a very narrow gap between the two sides of the image so that I only have to make one cut with scissors to separate them. Be sure to leave some blank space off to either side of the pair for joining the sheets together later. Consider putting several copies of the art on each sheet to avoid reprinting if you should fail transferring the art. It will happen, it just happens less with practice.

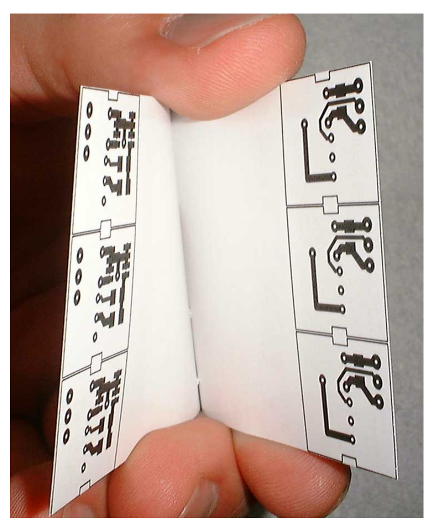
Printing the art:
Crop the image manually to avoid autocropping should your file be larger than the maximum printable area of the printer. Be sure to set the resolution for the image to match that of your original art. If there is a mismatch, the printed artwork will be off scale and parts will no longer fit on the defined pads.

Set the printer to maximum darkness for the best toner coverage, and turn off any image enhancing modes like PQET. Those enhancements only serve to blur and distort the image. Print the image on the glossy side of the photo paper. Print a marked test sheet to determine the paper path of your printer. I keep a post-it note on the printer indicating the proper orientation of the paper in the input tray.

Preparing the transfer sandwich:
Cut out the artwork on three sides, leave no hangover. Make sure you cut out the corresponding three sides of each copper plane, otherwise there will be no way to join the pieces for double sided boards. Leave as much of a tab on the fourth side as is comfortable. I prefer to put the tab in the longest dimension of artwork.



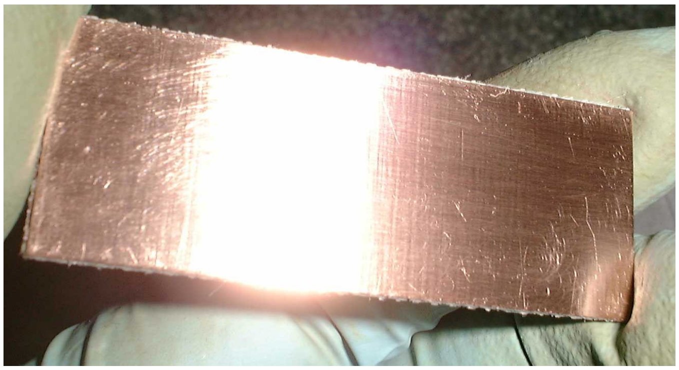
Here's where things get a little strange. Go to bed. Bring a stapler and a strong light with you. I have a swivel light with a 100 watt bulb over my bed. Lie down and turn on the light, keeping it right over your head. Hold the two halves of the artwork together, making sure the toner is on the inside and the pieces are in the same direction (ie, the proper direction to produce a circuit board). Hold the two pieces up to the light and slide them around until you have a good alignment between the top and bottom layers. DIP parts with pilot holes make for a great alignment array, and most of my projects use them anyway. Be sure to get as much of the board in as good of an alignment as possible. It should be sufficient to see light through the pilot holes.



When satisfied with the alignment (it'll take a while to get it just right…) pinch the two pieces together so they don't slide. Staple the tab close to the edge to lock the pieces together. I use as many staples as is practical.

Preparing the circuit board:
Cut the circuit board to size before you begin.

Procure a plastic tray with walls about two inches high, wear thin gloves, have a few pads of steel wool (00 is good), and have acetone and paper towels handy. Clean a piece of ceramic tile or countertop thoroughly, and set your clothes iron to maximum heat and minimum steam. Put the circuit board in the plastic tray and scour it with the steel wool. I tend to go in one direction, and then switch 90° every now and then. The idea is to remove all oxidation on the surface of the circuit board and provide a regular surface structure for the adhesion of the toner when it's melted. Get rid of as much discoloration on the board as is possible.



After the board is very bright and shiny, clean it off with the paper towels and acetone. This removes the iron and copper particles, but also removes any skin oils that will oxidize the metal more quickly than it would out in the air alone. Note that you could use a strong acid to clean the metal, but that doesn't put in the nice surface geometry that sticks to the toner. Some guides caution against using steel wool as it may cause "rusting" of the copper in the long term. That's an unreasonable claim. Copper is fairly hard and the weak steel wool particles are not likely to embed themselves in the surface. The acetone and paper towel remove most of the residue. Finally, iron particles would not survive the etching bath. They would be turned to trace quantities of ferric chloride and dissolve in the solution.

Transferring the art:
Fill a basin big enough to hold the circuit board with piping hot water.

Slip the artwork sandwich over the shiny circuit board, using the cut edges to align the art on the board itself. Since you cut three of the four sides, this should be trivial. Set the work piece on your surface (the ceramic tile or counter top) and use the iron to heat the board. Start on the stapled side of the art and slowly glide towards the loose end. This tacks down the art and prevents it from shifting. This process should take about thirty to forty seconds. Starting on the stapled side also reduces misalignment due to heat expansion in the joined side.

Start again on the stapled side, but this time put the iron flat on the circuit board. Avoid the steam vents and use a flat part of the iron. Lean down on the iron for fifteen seconds to ensure heating and pressure. Move the iron over, and repeat. Do that until you reach the end of the board. Once completed, go around the edges of the board with the tip of the iron. Go slowly.

Flip the board over gingerly. Don't grab by the tab or you'll disturb the alignment of the pieces. Move quickly to avoid being burned. Otherwise, use a heat resistant glove. Repeat the pressing process for the other side of the board. After both sides have been tacked, you may use the tab to move the board.

At this time I'll reiterate the importance of a clean tile and clean iron. If there is any debris on either surface, it'll scratch up the toner and paper and prevent clean transfer. Often it'll cause bubbles or pits or fragile spots in the toner transfer. Once completed, throw the board into the hot water bath to solidify the tone, but with minimal warping of the copper and paper sheets. Cold water works, but tends to cause rapid shrinkage that cracks the toner.

Removing the paper:
Let the board sit in the basin until the paper is translucent. The time required depends completely on the size of the board and the type of paper. A half hour is usually sufficient, but is often overkill.

Once the paper is soft from soaking, move under a faucet and run a slow stream of hot water. Put the circuit board under the stream and use a soft or medium toothbrush to remove the paper. Figure eight or circular motions work best. Linear motions are slow. Don't be too aggressive. Use the pads of your fingers to rub away complicated or stubborn areas. You're much more sensitive than the toothbrush, and are better able to prevent damage to delicate arrangements of toner. This is the most tedious part of making the circuit board.

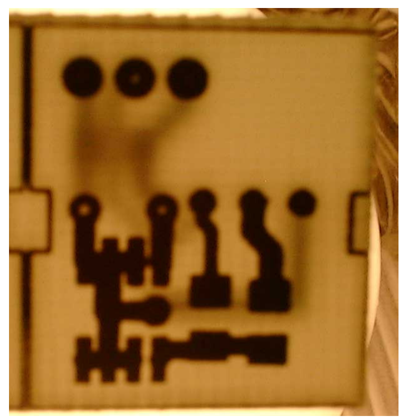
Don't try to remove all the paper, especially over the toner. Only the copper needs to be exposed. Excess paper on the toner only serves to strengthen the resist.

Fixing transfer problems:
Sometimes toner will come up when you remove the paper. Small fixes can be done with a permanent marker. Dab, don't draw. This puts on more ink. Also, the solvent in permanent markers will dissolve toner. Dabbing prevents smearing. There are some ethanol based markers that don't dissolve toner, but they're difficult to find because they're uncommon and unlabeled.

For wider errors or errors that involve a larger area, use electrical tape. Use your old ceramic tile and a hobby knife to cut out pieces of electrical tape to do any patch work. Use a flat object (such as the side of the knife) to squeeze the patch onto the board.

At this point, you can still abort and try the transfer process again. Acetone is easy to get and does a wonderful job of removing toner. Most organic solvents (ie, toluene) work well too, but are more expensive. If you don't have a paint store handy, household cleaning products like Goof-Off or Goo-Gone work well. Use gasoline if you're desperate.

Etching the circuit board:
Prepare two trays big enough to hold your board. Try to keep the size as small as possible. Fill one with water. Put the circuit board in the other. Wear gloves, an apron, and do this outside. The etching solution will destroy clothing and burn skin. Pour on hydrogen peroxide and hydrochloric acid. The ratios depend completely on the concentrations of the chemicals used. Do the math for your concentration. The hydrogen peroxide oxidizes the copper and the hydrochloric acid neutralizes the metal oxide base, resulting in copper chloride and water. Don't breathe the fumes. They're painful, so you probably will avoid breathing them of your own accord.



Swirl the board around in the solution, flipping it over now and then. Depending on the concentrations of the chemicals used, this could take a while, or just a minute or two. Be sure to wear gloves. Apron and goggles are good too. You may have to add chemical during the process. If the board is dark, but the darkness isn't dissolving, add more hydrochloric acid. If the board is bright but nothing is happening, add more hydrogen peroxide.

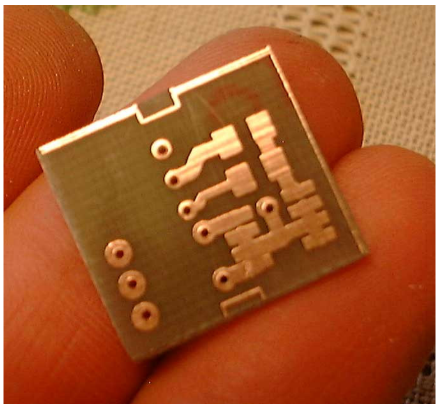
A note about the ferric chloride method: It's dismally slow, expensive, and hard to regulate. The solution is rather opaque, and so it's easy to over etch the board. The hydrogen peroxide and hydrochloric acid method produces a clear solution that allows for precise removal timing. I often use 15% hydrogen peroxide and 31.5% hydrochloric acid. The solution etches boards the size of my microcomputer (pictured at the end) in about three minutes.

Dispose of the solution in accordance with your local regulatory agency. In many areas, it may simply be diluted and flushed down the drain. Ferric chloride is an active organic catalyst and harms many ecosystems. Hydrogen peroxide, hydrochloric acid, and copper chloride are much friendlier for disposal.

Fixing etching errors:
Bridges are easy to fix. Use a hobby knife and make two cuts around the bridge in a V shape. Remove the sliver of copper.

Breaks are somewhat more difficult. I use a very narrow gauge of wire as a jumper, and solder it down to the trace as tightly as possible.

Drilling out holes:
I have less experience here because I haven't purchased any specialized equipment for the task. If any volunteers would like to contribute information, it would be very much appreciated.



A simple drill press and wire gauge bits seem to work well. Be sure to change the speed of the drill press, as the usually come from the store in the lowest speed. I use the maximum speed of my press, around 3200 rpm. The slower speeds are good for wood, but when cutting metals it does more tearing and less cutting.

Also, be sure to chuck the bit as high up as possible to reduce any bending of the bit. This will help keep the bit cool, sharp, and intact. More exposed bit means more bending which means more breaking. Pilot holes etching into the board make drilling a lot easier and more precise. Use a wood block under your circuit board to provide an easily movable stage and a sacrificial backer to preserve the bit.

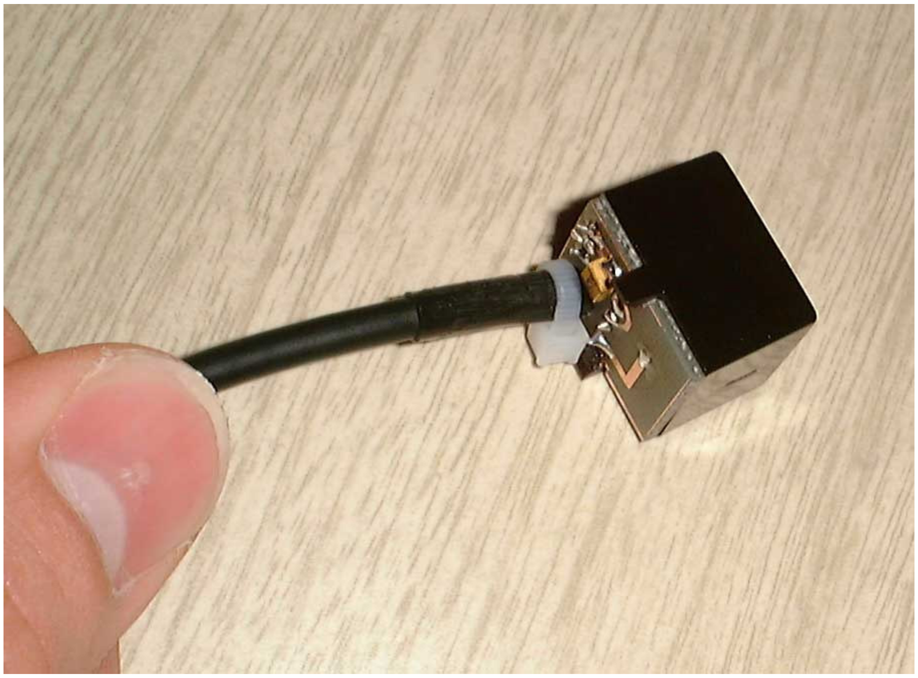
Wire gauge high strength bits are good for this purpose. There are specialty PCB bits, but they're hard to come by and rather expensive. They also require much more specialized equipment to use. 74 gauge holes make for good vias. 64 gauge holes fit most DIP parts. Experiment with the bits to find sizes with which you are comfortable.

Connecting vias:
Very small gauge pre-tinned wire is handy for completing vias. Cut a long length off your spool and strip all of the insulation. For each hole, insert the wire and bend a small (1/16th of an inch or so) tab onto the copper on one side. Place the board flat and bend the other side of the wire flat with the copper. Use a hobby knife to snip the wire. A very sharp knife will cut the wire but not damage the copper. The knife is also good for making hard bends that lock the via in place. Solder them down. I work in batches to speed up the process – laying down several vias and soldering them in groups.

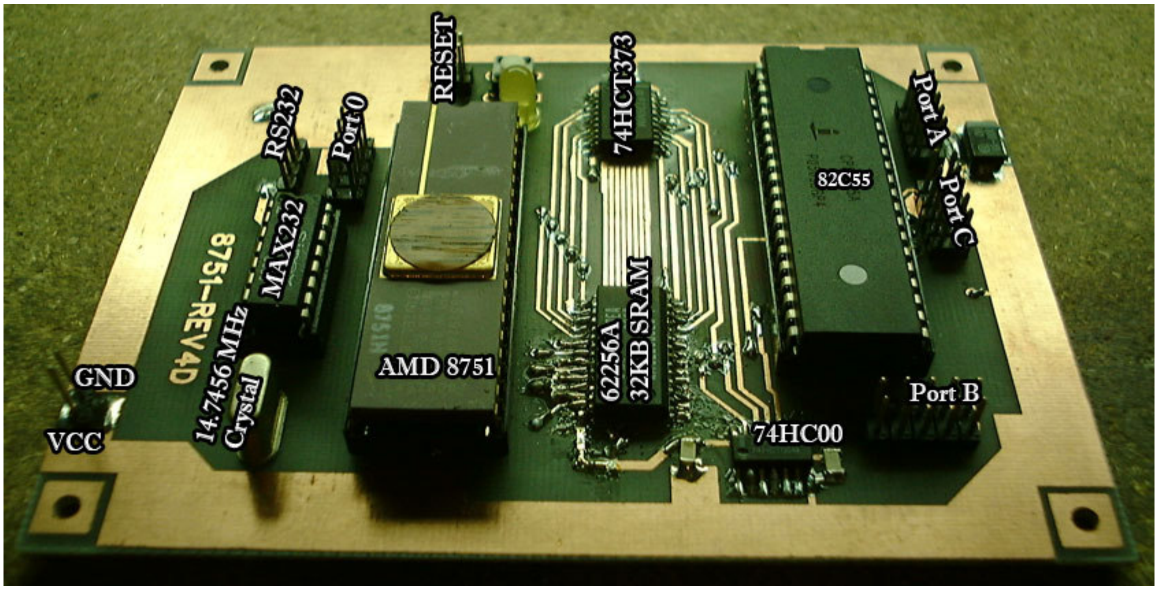
Soldering surface mount parts:
Tin the pads before attempted to solder the parts, but avoid leaving large bubbles of solder. For IC's with many pins, I like to place the part on the pads and solder opposite corners to tack the part down, and then solder down the rows. Smaller parts like surface mount capacitors may be held down with tweezers and tacked down. Soldering technique is beyond the scope of this article, but is mentioned for completion.

Notes about the pictures:

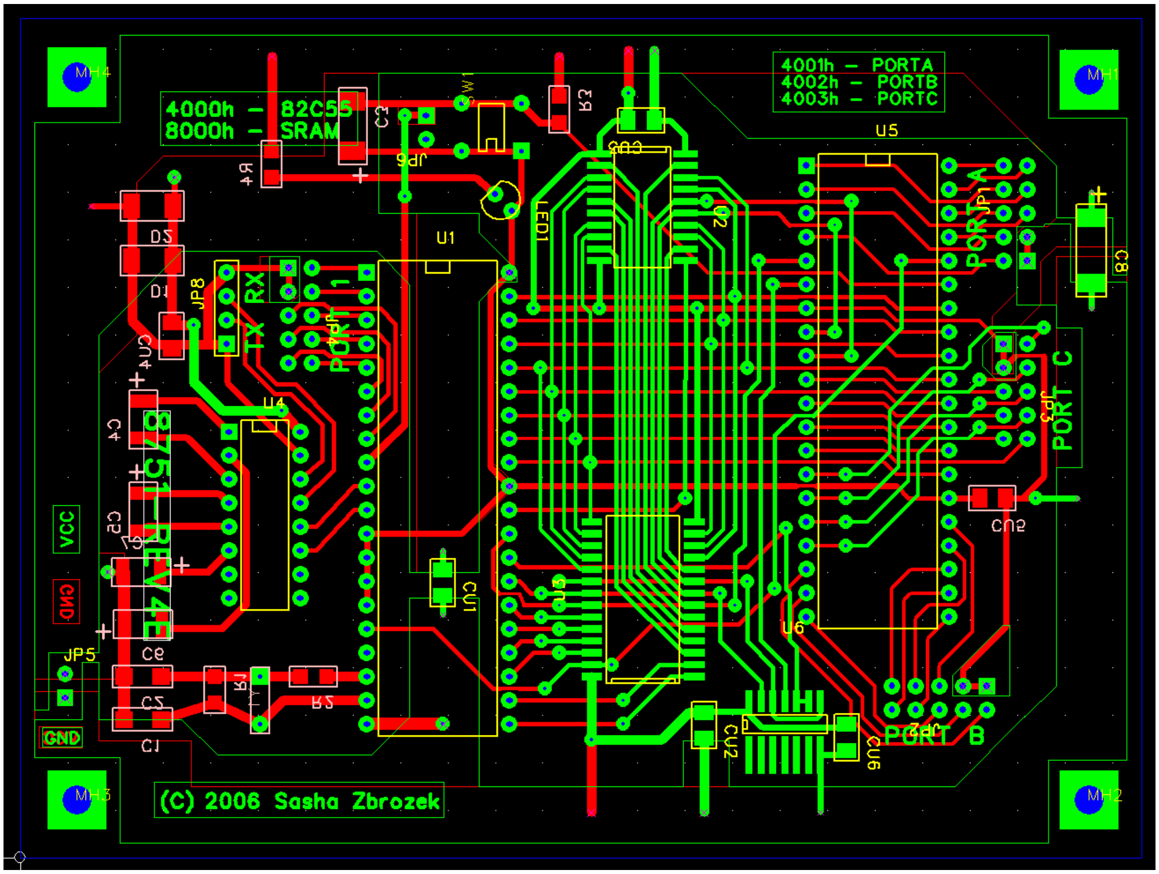
These images came from a small project as part of my MythTV system. It's an RS232 attached infrared receiver. The schematics are available on www.lirc.org in the serial receivers section. Here's the finished product:



And here's an example of what can be produced with this homebrew PCB method:



Since taking this picture, I've switched out the processor and crystal in favor of a 22MHz, flash based set. Here's the design of the board:



Thanks, Allan! FreePCB is wonderful software!