# 12V Laptop Charger

This little project was mostly designed to re-acquaint me with the PCB design and etching process, as I have not done one for many years. I have a need for a number of boost DC-DC converters for running tools, laptops & battery chargers in my work vehicle. A number of friends would like similar items, so I made the design universal.

Below is the schematic of the circuit based on the Linear Technology™ LT1170. The LT1170 has a 5A maximum switching current, LT1171 (2.5A), and LT1172 (1.25A). It will take about 40V in and produce up to 60V out. It can do more if external shunting is used, but I wanted to keep the design simple & cheap. A large laptop running and charging will pull as much as 5A, so I used the LT1170. It will internally limit its maximum switching current, thus it is almost indestructible. I have included the option for a crowbar zener & diode to short the output should it go above a set voltage. This should blow the supply fuse, which should be sized to allow this. I used a 20V zener & diode to set crowbar to about 20.6V, as the laptop was rated for 19.5V.
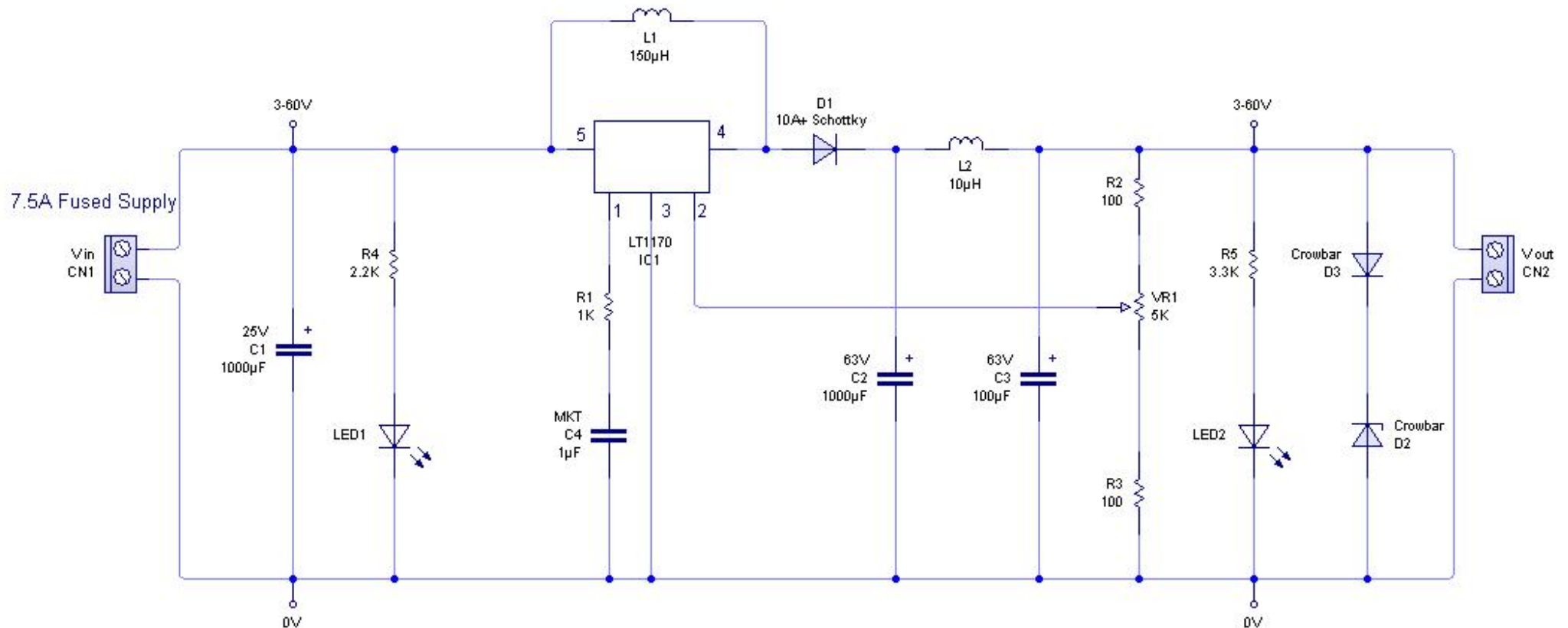
Once the design was complete, it was converted to a PCB layout. Although the software I used can lay out the PCB automatically, it tends to need a lot of manual adjustment to get things where you want them. For example, the LT1170 has to be as close to the edge of the PCB as possible to allow it to be attached to a heatsink. Below is the PCB layout after a lot of work. Also the TO-220 5 pin package was not in the library so I had to fiddle and make one.
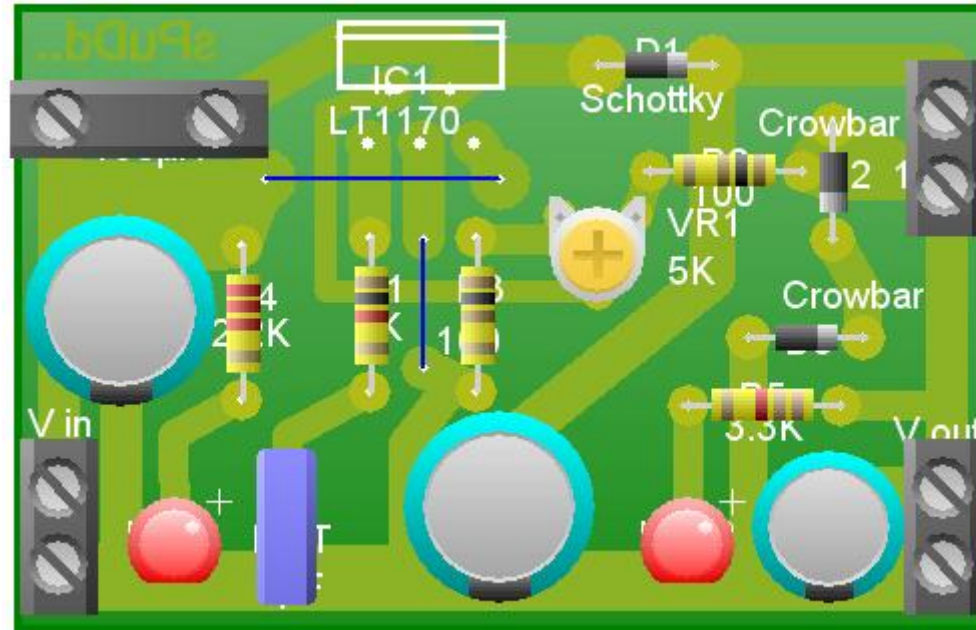
I bought some PCB Press & Peel, which is fed into a laser printer. It took me a short while to find a way to move the PCB design around the page and keep feeding the same sheet back into the printer over and over again. This makes good use of the sheets. Later I found I could copy/paste the image onto the same page as many times as I liked, thus enabling me print once with as many designs as would fit the one sheet of paper. This is much more efficient use of the transfer paper. Oh, and I had to turn off the Header & Footer notes in the page settings. You can see this problem at the bottom left of the sheet in the picture:
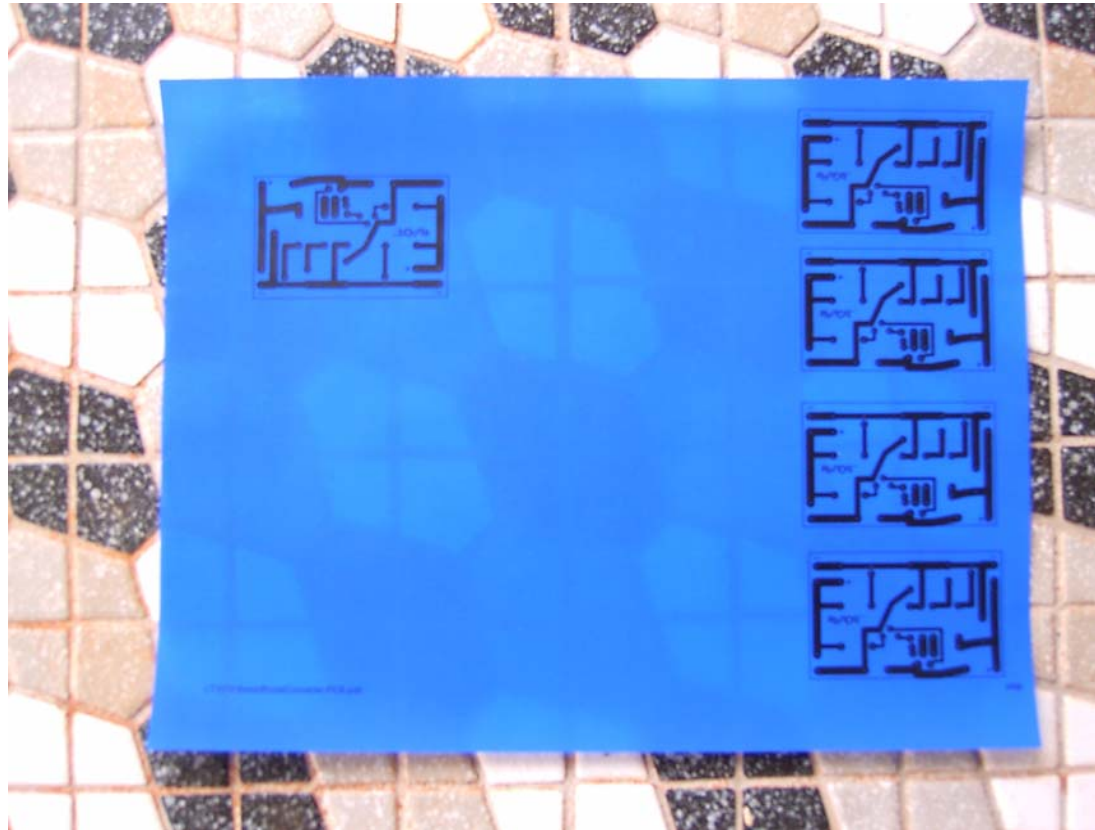
I measured up the PCB layout and cut a full strip of mini boards from a 300mm x 300mm sheet of blank PCB material. They just need to be deeply scored on both sides with a strong utility knife, and then snapped. They can also be cut with a hacksaw, but the knife tends to produce a very straight cut.

Once they have all been cut out, run a small file over the edges to remove all the burs. This is essential as burs will affect the transfer process. These should only be cut out just prior to an etching run, as your fingerprint sweat will corrode the copper very quickly. Once you are ready for the transfer process, clean the copper with fine steel wool under running water, or a 1000+ grit foam sanding block from a hardware store. Then dry them with a clean lint free cloth. Just prior to placing the transfer on top, wipe the PCB clean with pure alcohol. How clean the PCB is makes a HUGE difference to the transfer process.

Then cut out the PCB transfers with at least 20mm of overhang each. I found that if I held the PCB up to the light, I could centre it on the transfer. Then I would hold it down and apply the tip of the clothes iron to a section of the PCB. This made the transfer stick to the PCB long enough for me to tape a sheet of blank paper over the PCB. The paper helps the iron slide over the film without catching. Its essential with tiny PCBs, or you will burn yourself plenty. Some sticky tape can also be used to tape the edges of the transfer to the desk. Its important that the transfer not move about on the PCB, or it will smudge.
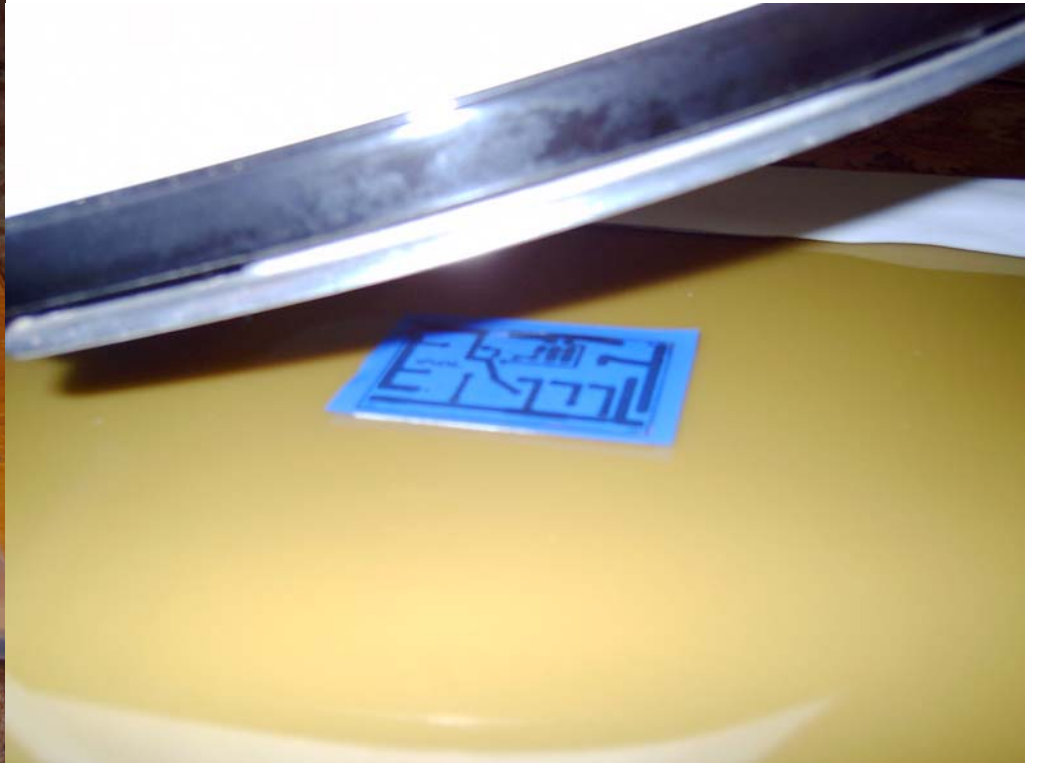
I found that the iron I had needed to be set to maximum. But even so, the thermostat only came on every 2 minutes or so for about 5 seconds. So although the temperature was high, the amount of heat lost to the PCB was small. I kept moving the iron around slowly with firm pressure until the PCB left a full imprint in the paper. It took about 5 minutes for each mini PCB.

You can see the PCB on the left is missing some of the top most track. This was my first PCB, and it was not cleaned properly. The later ones turned out better. I did learn that I need to allow a couple of mm border around the whole PCB, as the edges are the hardest to get the PCB transfer to stick too. Later after etching, you can trim off the extra border with a file.
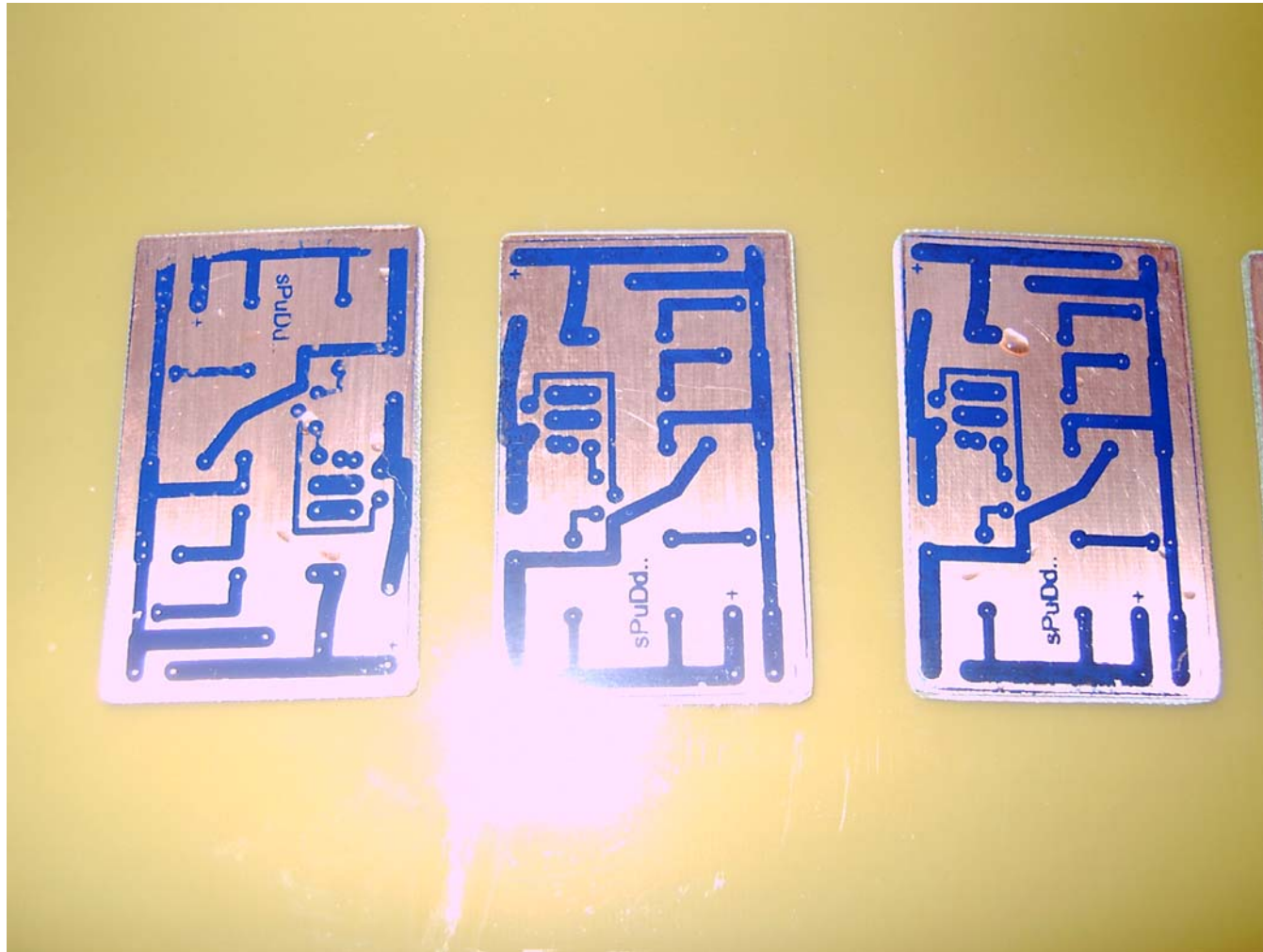
As we are dealing with Ammonium Persulfate for the etching chemical, a deep stainless steel laundry sink is an ideal place to setup the etching tank. Any spills can be contained and treated with water easily. I boiled the water to about 70C. A fish tank air pump makes the process move quickly. The five PCBs were done in under 10 minutes. Drill a hole in each PCB and attached a short piece of wire, so you can lift each one in and out of the tank without touching any chemicals.
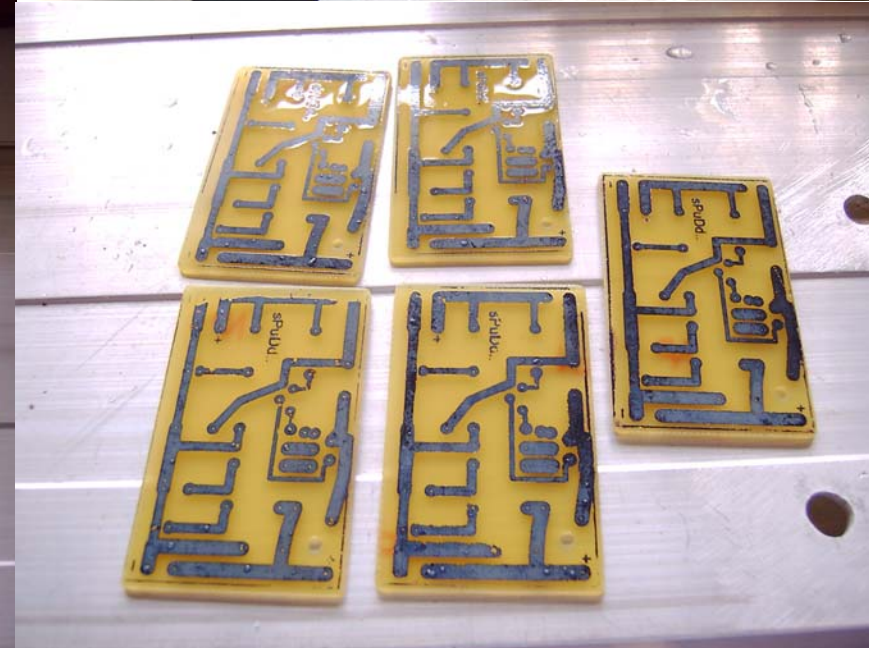
The PCB is then drilled. A mini drill like the Arlec or Dremel is the go. Most holes are 0.5mm, 0.8mm and 1.0mm and need to be fairly accurate to allow each component to line up properly. Larger items like the screw terminals used a 1.0mm bit. Leave the PCB transfer coating on until you are going to use the PCB. The copper will tarnish quickly. I cleaned up one PCB ready for immediate assembly. I found the 1000+ grit sanding block the best at removing the toner, and it also cleaned up the drill hole burs.
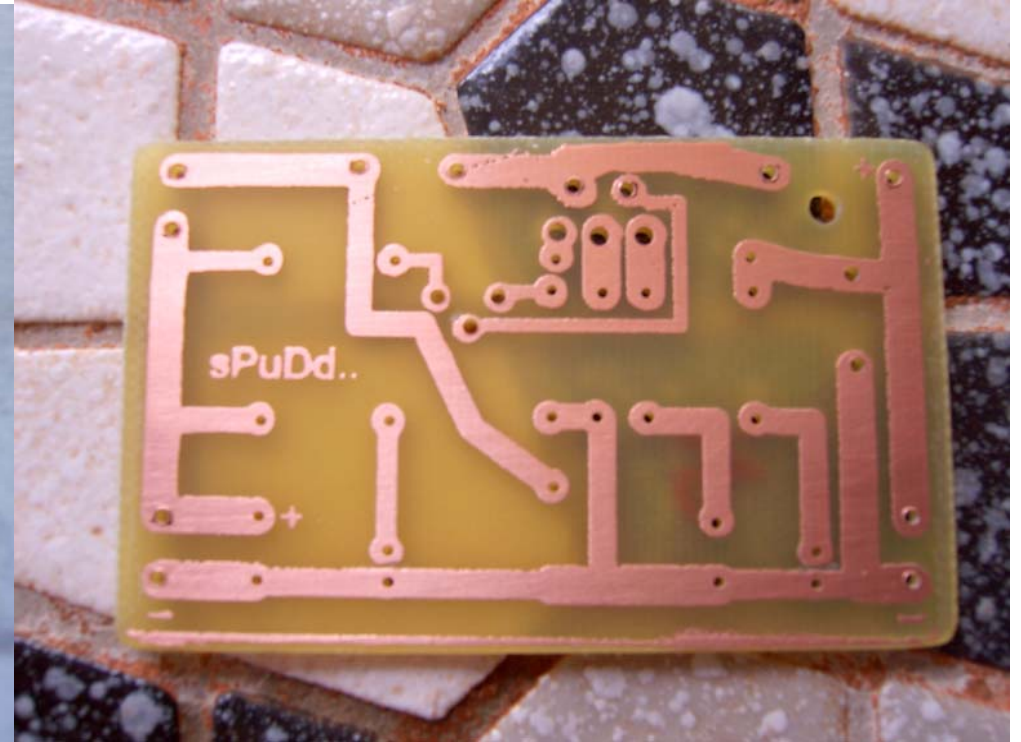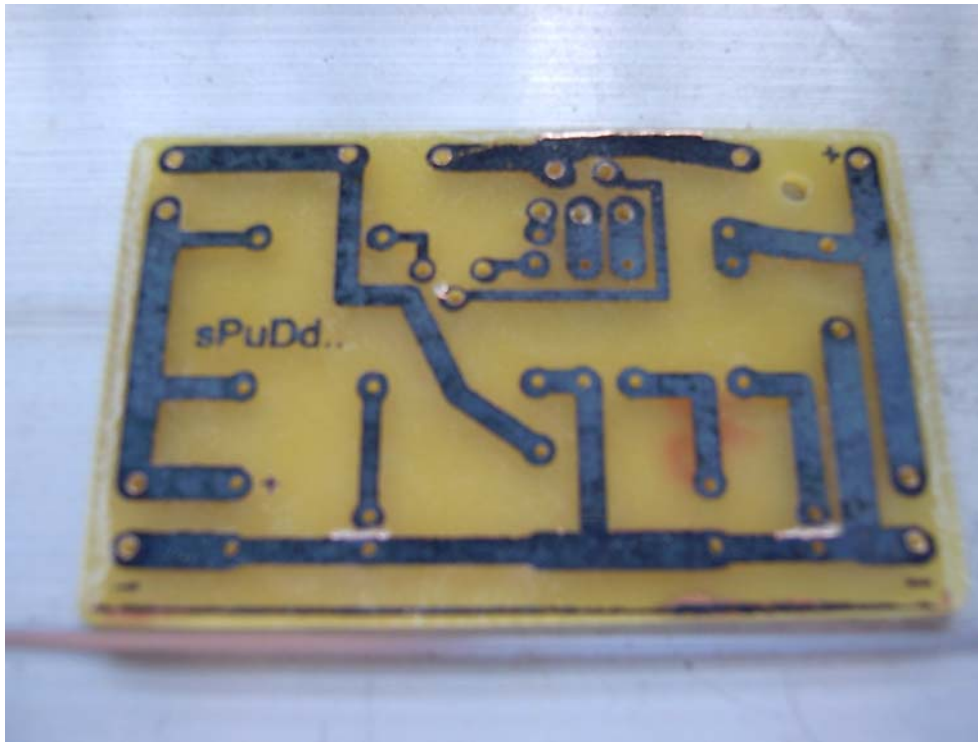
And here is the finished loaded PCB, tested & ready to go.
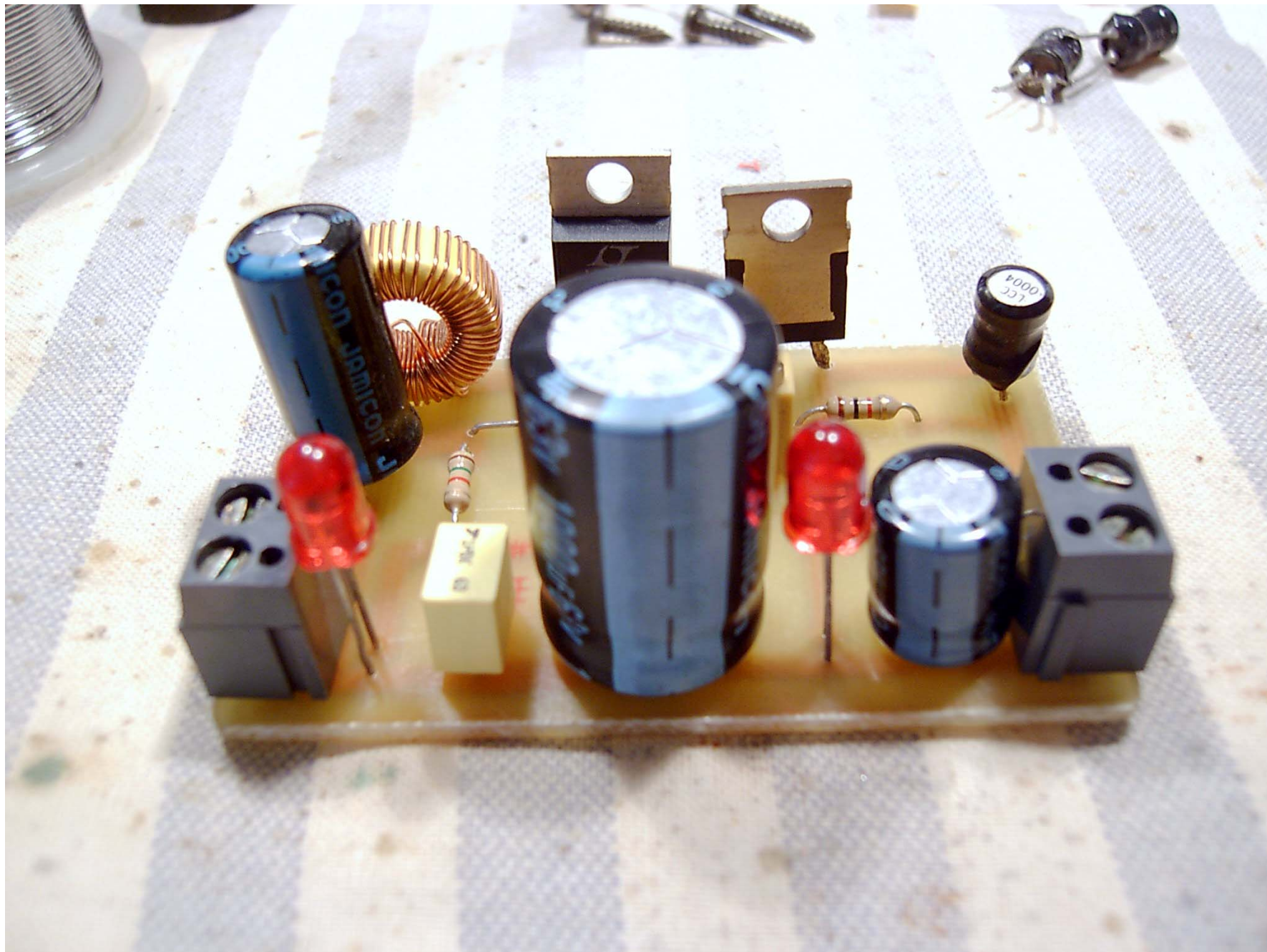
I coated all the tracks with solder. This improves cooling for each component, increases track current capacity, reduces dry joints and fixes any hair line cracks that may be in the etched tracks. I prefer a 60/40 lead solder with synthetic flux. Lead free solder causes tin whiskers. That's one for you to Google. Last thing to do is clean the solder resin off the PCB with alcohol and give it a coat of PCB lacquer to prevent corrosion.