Printed Circuit Board Fabrication for the Hobbyist

When one gets involved in all of the various aspects of the electronics hobby, as I do, the time will eventually come when it becomes necessary – or at least most convenient – to use a printed circuit board (PCB) instead of "dead bug" point-to-point wiring of a project. There are several options open to the hobbyist when it comes to making custom PCB's for any given project. These range from hand cutting the necessary voids in a copper-clad board, to printing and etching a board using the toner transfer method, to one of the ways I made boards years ago, where resistive pads and traces are applied to a copper clad board which is then etched, to having custom boards professionally manufactured by a PCB house. Let's take a look at some of these methods one at a time – you might find one that suits your needs.
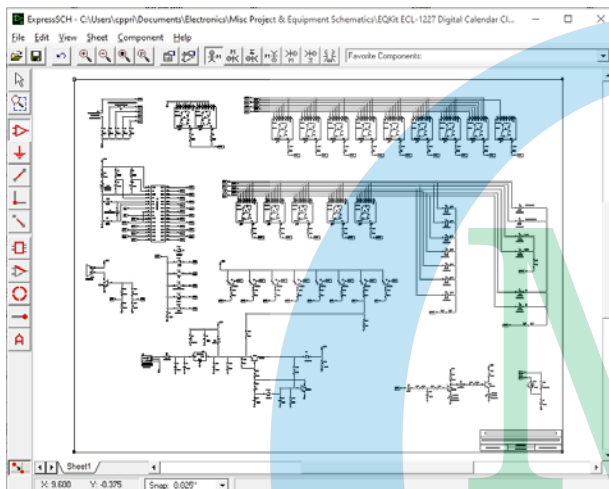

Figure 1 - The ExpressSchematic interface

I am going to start at the place that I always start – with a good, clear, and concise schematic. Now… there is nothing that says that the schematic cannot be hand-drawn, although my chicken scratching hardly passes for "clear" or "concise". As a result, I use a piece of freeware schematic creation software, of which there are several available. A quick Google search will turn up several results for consideration, including KiCad, Eagle, and ExpressPCB.

The software that I use is a package known as ExpressPCB (Figure 1), which includes the schematic creation component ExpressSchematic. This software includes numerous component definitions, but also provides for the creation of definitions of custom or unincluded components. Once all of the necessary component definitions are at hand, it is a simple matter to draw out the schematic to match the design that is in mind.

Translating the schematic to a PCB design is quite another story. In a schematic drawing, it is quite acceptable to have lines cross each other. However, in the layout of a PCB, the traces cannot cross each other *on the same layer.* Therefore, either some more complex routing of the traces is necessary, or else the board can be made as a two-layer PCB, with traces on each surface of the PCB. In this case, when a trace has to be shifted off from one side of the PCB to the other side in order to cross another trace, the connection from one surface to the other is made using small plated through holes called *vias*.

Of course, if a trace needs to be on the second side of the PCB, generally the upper surface, it can be routed there from its origination at a component lead connect point. In such a situation, it must be remembered that the through hole for that component lead will have pads on both sides of the PCB.

Under general through-hole component circumstances, most of the traces are placed on the underside, or lower surface, of the PCB, known as the *foil side*. The components are installed to the upper surface, known as the *component side* of the PCB. Sometimes, the positive voltage rail is routed on the component side of the board and the part of the foil side not occupied by

current-carrying traces is filled by a copper plane to which all of the common, or chassis (ground) connections are made. Care must be taken to avoid placement of such a ground point in an "island" of the filled plane that does not have adequate connectivity to the rest of the plane.

I use the ExpressPCB (Figure 2) software component to lay out my PCB's, regardless how they are eventually produced. A few words are in order here regarding ExpressPCB. The software is a free offering of *ExpressPCB, LLC,* a company that is in the business of manufacturing printed circuit boards for their clients. They make the software available at no cost in the hope that the user will also use their board-production service. Unfortunately, I find their pricing to be prohibitive, especially as compared to some of the other board houses that I have used in the past. I will talk some more about that later.
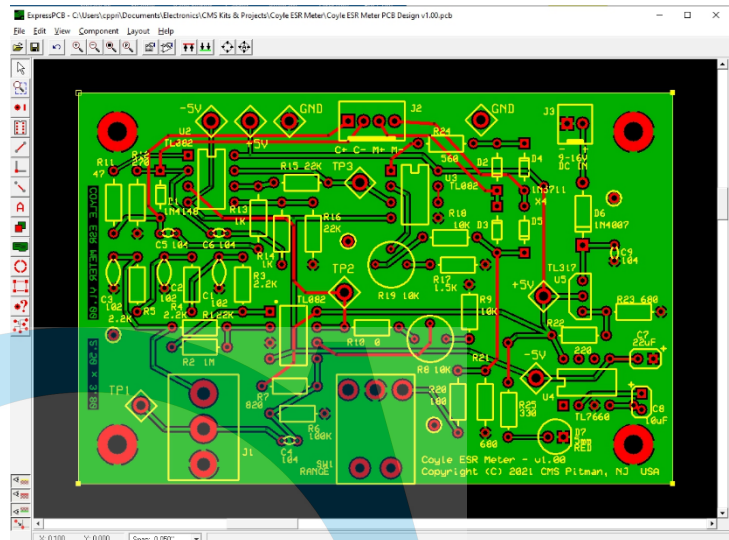

Figure 2 - The ExpressPCB interface

For home-shop manufacture of PCB's, we start with a sheet of copper-clad FR-4 (Figure 3) or phenolic board. If we are making a single-sided board, we use single-clad board stock as the base. If the board needs to be double-sided, we use double-clad board stock. The thickness of the copper layer on the board is described in ounces, meaning how much one square foot pf the copper material weighs in ounces. Most standard PCB's use one-ounce copper. A one-ounce copper layer will have a thickness of ≈35μm or 1.4 mils (0.0014"). Two-ounce copper will be appropriately thicker at ≈70μm or 2.8mils (0.0028").
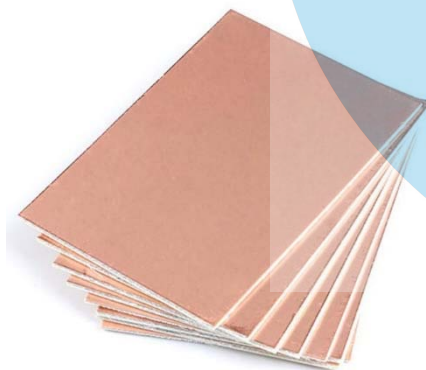

Figure 3 - Copper-clad FR-4 boards

When the board is produced, we begin by covering the wanted part of the copper layer – the traces and pads – and then immersing the entire board into a caustic solution that will remove the uncovered copper, leaving only the covered and therefore protected areas behind.

Now would be a great time to talk a little bit about safety. The processes involved in the production of printed circuit boards involve the use of chemicals, many of them caustic and most of them quite pungent and even harmful to inhale. Be sure to wear protective clothing and PPE including but not limited to chemical splash goggles (Figure 4), a rubber apron, and rubber gloves. Also be sure to work in an environment with proper and adequate ventilation. Finally, be sure to have neutralizing agents on hand, *e.g.,* vinegar when working with alkalis and baking soda when working with acids, so as to neutralize any spills or splashes, especially on your person. I will not repeat these safety instructions at each and every step of the way through this article, but they apply

Figure 4 - Chemical splash goggles

none the less. Be aware of the hazard and proceed at your own risk. Also, be sure to always properly dispose of the used chemicals following the applicable hazardous waste requirements.

To actually produce the PCB's, we have to go through a series of steps, depending upon which method of production has been chosen. I will start out by describing the old peel-and-stick method first.

Once upon a time, before the common use of laser printers and dry toner type copy machines, it was a commonplace practice to create PCB designs using a series of stick-on labels that would cover and thus protect the copper on a copper-clad board. These labels were available as traces and pads in various sizes and shapes (Radio Shack catalog number 276-1577, Figure 5), which would be placed onto a prepared copper-clad board and then pressed into place to seal them to the board surface. The design was built up slowly, selecting each trace line, angle, dogleg, and pad from the master sheet in turn and moving them into place on the board, overlapping them as necessary to create a continuous path of protected copper for each trace, to and into its connected pads. Some brands of these labels would have the center holes of the pads already punched, so that when the pad label was put in place and the board was etched, a drill center would be apparent on the center of each pad. This was important as a means of keeping the drill from "walking" on the board when drilling the component lead holes later.

Figure 5 - Radio Shack 276-1577

Once the design was complete, I would use a standard baker's rolling pin with a sheet of glass under the copper-clad board to tightly seal the labels to the copper-clad board. A tight seal to the board is necessary to keep the caustic solution from bleeding under the label and deteriorating the pads and, more importantly, the traces. With enough bleed, a narrow trace could potentially be bisected by the caustic creeping under the label.

Figure 6 – Ferric chloride

Once the labels were all sealed to the board, the board would be immersed in a tray of caustic solution. Back in the bad old days, we would use ferric chloride ($FeCl_3$) in a shallow plastic pan or tray, using enough of the $FeCl_3$ to completely cover the board in the tray. With the board submerged copper side up (I only made single-sided boards back then), I would rock the pan forward and back gently to wash the $FeCl_3$ over the surface of the board. In about ten minutes, the board would be done, with all of the exposed copped removed. The copper was generally etched from the outer edges of the board inward towards the center by this method. Once the board was fully etched, it would need to be rinsed under cold running water to halt the etching process. After that, the protective pattern labels would have to be peeled off, usually with the help of a hobby knife. Then, the requisite holes would have to be drilled in the center of each pad, to accept the component leads. This is best done in a drill press, as it is quite difficult to drill these

holes accurately with a very small twist drill in a portable drill motor, cordless or corded.

Making PCB's using this method had one huge drawback. Disposal of the used $FeCl_3$ was a problem, and as a second negative, the solution would weaken as the copper was dissolved into it, making the process take progressively longer and making disposal of the solution even harder to manage. There had to be a better way, and I found it.

I developed a plan whereby I would, wearing gloves, of course, put about a tablespoonful of $FeCL_3$ into a small sponge, and then would wipe the solution onto the copper-clad board, working from the center towards the edges, and I would actually see the copper wiping off the board in a matter of a minute or so. When etching was complete, I would rinse the board and the sponge under cold running water, and then I would store the sponge, after it dried, with the bottle of $FeCL_3$ for the next use. On occasion, I even rinsed off the gloves and re-used them. Waste not, want not!

This method worked well and served me for quite a few years. I used to buy the $FeCL_3$ right at my local Radio Shack® (catalog number 276-1535 – Figure 6), where I would also buy my one-ounce copper-clad board stock. However, as time went by, I was always looking for the next better idea, and I found and fooled around with a few of them over the years. The next method was the photo-resist method.

This method also involved etching the board, but the board preparation was done differently. In the peel-and-stick method above, I prepped the board by scrubbing it with steel wool and some dish soap, blotting it dry with a paper towel *and then not touching the copper surface at all with my bare fingers.* This was because the skin oil would prevent the labels from adhering properly to the copper-clad board surface.

With the photo-resist method, the board was prepped initially following the same procedure. However, and this is quite important, it there are any burrs or raised edges on the board from saw-cutting the material, be sure to file and/or sand them off. The edges must be at least flush with the surface of the board for this method to be successful. This is because any lip on the edge of the board would affect the entry of light to the board during exposure. There are two alternatives in this method as regards the board stock. One is to purchase pre-coated boards that already have the photo-resist material applied (Figure 7), and the other is to apply the photo-resist paint to the board yourself. If you choose to coat bare stock yourself, it should be done with a spatula rather than with a brush, so as to get a smooth and even coating across the board without brush tracks

Figure 7 - Photo-resist board

in the paint. You may need to thin the paint before you can get it to flow properly. Use acetone for this purpose. If you thin the paint too much, stir it for a few minutes; it will thicken as the acetone flashes off. Allow the photo-resist paint to fully cure on the board before continuing with the process. It helps to heat the board slightly to aid in the curing of the coating. I have done this using a re-purposed Mr. Coffee™ heater plate with the water tank and upper portion of the coffeemaker cut off, and a solid aluminum disc 1/4" thick cut in a round to fit onto the heating plate and bring its surface up above the edge lip of the heater plate. The coated copper-clad board was placed on top of this aluminum disc and left in a darkened room overnight. It is

important to keep the photo-resist paint from being exposed to light as much as possible, and especially to bright or UV light.

The next step is to prepare your photo mask. This is done by printing, on a clear acetate sheet, a *negative* image of the PCB trace and pad layout at its full and exact size. Be sure to print this with zero scaling applied, and *mirrored* at that full size. I prefer to take the image out of my design software as a .PDF file, which I then print mirrored at full size on a 3M transparency sheet. I also set my printer to outfeed the printed sheet directly out the back instead of the printer rolling it over and out-feeding into the top tray, as this prevents fine split lines from appearing in the still-soft toner on the acetate sheet as the sheet is tightly curled in the rolling process.

This print, generally cut out of the full sheet, is then placed into a glass sandwich stack made up as follows:

1. the photo-resist coated copper-clad board on the bottom, coated side up;
2. a sheet of 1/8" window glass goes on top of the copper-clad board;
3. the mask print goes on top of the glass *with the toner side facing downward;* and
4. a second sheet of 1/8" window glass goes on top of the print.

This entire stack is then exposed to UV light, preferably in a dark environment. I have at various times used a home-made exposure box using a cardboard carton and a black light bulb in a lamp base or socket, a similar box with a "black light" fluorescent tube, and again a similar box with a regular fluorescent lamp. I have even tried it without the box, just placing the stack directly under a fluorescent lamp, within about ten inches of the lamp. I any event, the exposure will take at least fifteen minutes and the stack should be rotated a couple of times to even out the exposure. I used clear tape to hold the whole stack together so that the components would not shift during rotation, and I used a repurposed game board turntable to make it simple to rotate the stack – just turn the turntable 120° or so every five minutes.

After the exposure, the next step is developing. In this step, the board is immersed into a developer solution, typically a mild sodium carbonate (Na_2CO_3) (Figure 8) solution in water. A solution of about 1/8 ounce by weight to one gallon of water is appropriate. This equates to about 1 gram per liter of water for our metric-centric folks out there. A stronger solution runs the risk of removing the exposed photo-resist coating while washing

Figure 8 - Sodium carbonate

away the unexposed portions. We want the unexposed areas to be washed clean while the exposed areas remain in place for the etching process. If necessary, replace the developer solution midway through the development process, keeping the mix ratio the same. Do not increase the strength of the solution, no matter how much you may feel as if you need to do so. This process takes time; let it develop!

Next up after developing and rinsing is the etching process. This can be done with the ferric chloride ($FeCl_3$) as described, or it can be done with a solution of one part 30% muriatic acid ($HCl_{(aq)}$) (Figure 9) in two parts 3% hydrogen peroxide (H_2O_2). Be sure to add the acid to the peroxide instead of the other way around. Stir the solution gently with a plastic utensil to thoroughly mix the constituents, and then immerse the developed copper-clad board in the

Figure 9 - Muriatic acid

solution. Gently agitate the solution periodically until all of the visible copper has been etched away. The solution will likely turn a brilliant green color during this process, and that is perfectly normal. If the process slows down to a crawl before etching is complete, add a little bit of 3% H_2O_2 to the solution and continue the agitation. Once the etching is complete, remove the board using plastic tongs (which should also be used to handle the board while in the solution) and then rinse it under cold running water to stop the etching process. Blot the board dry with a paper towel.

At this point, the board can be drilled and the remaining developed photo-resist can be removed afterwards if so desired. However, I preferred to remove the photo-resist before drilling so that I could do a thorough inspection of the board. The photo-resist can easily be removed at this point with acetone, which can then be used to give the board a thorough cleaning. After inspection, lightly center punch the drill locations and drill the requisite holes for component leads as before.

The next method that we will discuss today will be the dry toner transfer method. This method involves the same etching process(es) as those already described, so that part of the process will not be described again. Nor will the board preparation, as that part of the process is also the same. What is different is how we cover and protect the desired areas of copper, separating them from the waste areas which we want to be removed from the board.

Of primary importance is the fact that this method requires the use of a laser printer or photocopier that uses dry toner. This process will not work with ink-printed images such as those from an ink-jet printer. However, an image may be ink-jet printed and then photo-copied onto the proper paper (as below), but the image may bloom some as the ink is absorbed into the paper, and there may be some scaling or size issues that crop up as a result of the print-and-photocopy process.

Figure 10 - Hydrogen peroxide

The difference is that with ink-jet printing, the ink is absorbed *into* the paper, while with laser printing, the print is adhered *on top of* the paper, and can be peeled off under the right circumstances. Anyone who has ever had a laser-printed sheet of paper stick to a vinyl binder and then leave its print behind will readily understand this.

Starting again with a computer image of the PCB design in its mirrored form, we want to print this design onto single-sided glossy paper having a weight of 130 GSM. The term *GSM* refers to *grams per square meter* and is a specification of the paper's thickness by weight for a sheet of a given area of one square meter. 130GSM glossy paper is approximately equivalent to a thickness of 110 microns, which in turn is equal to approximately 0.004", or the thickness of an 80-lb glossy text paper stock.

After we print the mirrored image on the proper paper, the toner is heat-transferred to the prepared copper-clad board. This can be done using either one of two common methods – the "iron" method or the "laminator" method. Both methods work, and both methods take some trial-and-error practice to get nailed down to a successful system.

The "iron" method involves using a standard clothes iron set at its highest temperature and *without* steam – in fact, pour any water that is in the tank out, just in case the valve doesn't quite work properly – to press the printed image onto the prepared copper side of the board. Using steady and even pressure, and keeping the iron moving so as not to char the paper, heat the print until the toner is softened and adheres to the copper. *Do not* attempt to peel the paper off the board; doing so will invariably pull the toner back off the board at least in spots. Just let the paper and the board cool for a few minutes, until the toner is fully set. We will pick up on this in a minute or so.

Figure 11 - Personal laminator

Meanwhile, let's talk about the "laminator" method. This method involves the use of a roller-platen type of laminating press (Figure 11), through which the board and the print are fed together, with the print aligned to the board with the toner towards the copper. Once the process has been perfected, a single pass may be adequate. Initially, at least, two or more passes through the laminator may be required. The variables here are the pressure, set by adjusting roller platen spacing, and temperature, adjusted via a thermostat on the laminator. These laminators typically cost about $70 USD and up, depending upon features and size. They are available from many online sources. I generally used a laminating pouch or folder to contain the board and the print when feeding it through the laminator. This is a necessity for smaller boards, which are not well handled by the rollers directly. Again, do not attempt to peel the paper from the board, as it will most likely pull the toner off the board, at least in spots.

Another alternative is the *Press 'n' Peel* system sold by Techniks of Ringoes, NJ. This is a blue emulsion-coated sheet onto which the mirrored image of the PCB design is laser-printed, and then heat transferred as above. However, in this case, the blue sheet readily peels away, leaving the toner on the copper surface. Go to https://www.techniks.com/ to learn more about the PCB manufacturing supplies offered by this company.

Once the heat transfer has been accomplished, immerse the board and paper fully into some cool water, and let it soak for a few minutes. Of course, this step is not necessary with the *Press 'n' Peel* system. The paper, once fully saturated, will come right off the board, leaving the toner behind on the coper. The board is now ready for etching, as described previously. After etching is complete, the toner protectant can generally be removed from the board with a paper towel wet with some 99% isopropyl alcohol.

One advantage of the toner transfer method is that you can put small holes in the pad masks to locate the drill centers. When the paper is soaked off the board and the board is still in the water, a *soft-bristled* toothbrush can be used to very lightly scrub the paper out of these hole locations, thus allowing the etchant to remove the copper at those points, making built-in drill centers for the eventual drilling process.

Another advantage of the toner transfer method is the ability to easily include text (**bold** works best) into the design and thus have it reproduced in copper on the board. Don't get carried away here. It is not nearly as capable as screen printing is, but it will support some basic text.

So how about the whole topic of screen printing? How can we get some semblance of that professional touch on our home-made boards? One easy method should already be evident – simply print a mirrored image of the desired text, in location on the PCB design, and heat-transfer that print to the completed board. Yes, the print will be black instead or white or yellow, and yes, it may wear off in time if the board is handled. But it *will* work at least enough to facilitate assembly of the PCB. Align the print carefully, and remove the paper from the board in the same manner as above.

Some other pointers for the amateur PCB maker…

1. When making the final drawing from which your resist mask for the toner transfer method will be printed, be sure to include corner markers at the proper locations. These are used to align the pre-cut and sized board exactly to the drawing.
2. When two-sided boards are being made, there are some other considerations to bear in mind
   a. The foil side of the board must be printed as a mirror image, but the component side print is printed without mirroring,
   b. A duplicated stack of glass and print must be placed under the board when using the photo-resist method, and then both sides must be exposed while maintaining proper alignment.
   c. When using the toner transfer method, again a stack of two prints and the double-sided board but be heat pressed, one print per side. Depending upon the heating method used, this can be done one sheet at a time (iron method) or all at once (laminator method).
3. When voids show up in your traces because of aggressive etching or etchant leakage under the mask, repair any such weak traces by laying fine wire over the trace and soldering it in place, using rosin-core solder.
4. Small defects in a trace or pad can generally be repaired simply by tinning the area with some rosin-core solder and allowing the solder to bridge the gap.
5. Practice! Practice! Practice!

OK… so much for rolling your own. How about having board(s) made professionally? It couldn't really be any easier than it is. While there are several PCB houses out there (*PCB house* is just another way of saying "PCB manufacturer"), there is really only one that I even consider using anymore. I have tried several different companies over the years, but nobody beats the speed, the cost, or the quality of the work done by JLCPCB (www.jlcpcb.com).

I am getting just a little bit ahead of myself here. Let's start at the beginning. In order to get PCB's made professionally, you will need to have a set of *Gerber files* for each different board ordered.

A Gerber file set is a group or related files (Figure 12), traditionally assembled into a single *.ZIP* file, that describe the board being ordered. The group is a set of files, most of which are numerically-coded text files, that lay out the specifics of the board in every way, shape, and form.
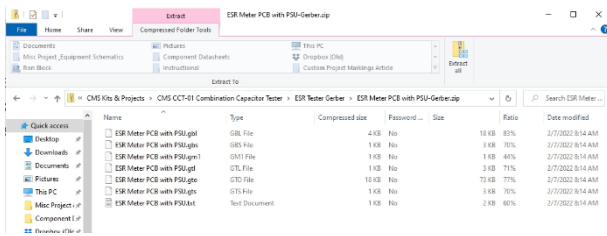
Figure 12 – Sample contents of Gerber .ZIP file

There will be files for each layer of the board, including the copper layer, the solder mask layer, the silk screen layer, and so forth for each side of the board as well as any intermediary levels on multi-layer boards. Finally, there will be a drill file that specifies the location and size of each and every hole to be placed in the board. The filenames of each of these files within the .ZIP file is pre-determined by the Gerber file standard, and those standard filenames are understood by the board maker.

Many – but not all – of the PCB design software packages out there will directly produce the Gerber file set as an output of the software. While the package that I use, ExpressPCB, does not output the Gerber file set, there is a company out there on the web who will take the .PCB file from the version of ExpressPCB that I use and convert it to a Gerber file set for a minimal fee. IN addition, if you were to order your boards from ExpressPCB, they will provide you with the Gerber file set as a part of the purchase price of the boards ordered.

It should be said that ExpressPCB is available in two versions, the so-called "Classic" version and the newer ExpressPCB Plus. The Classic version tops out at version 7.9 and has been "sunsetted" with support ending in 2024, though board ordering and production will be available as long as permitted by Microsoft. That's right… I said Microsoft. It has something to do with Windows support and security conditions. ExpressPCB, which is actually more capable, is up to version 3.2. Unfortunately, even that version does not output the Gerber file set, and my online source cannot yet convert the ExpressPCB Plus design files. As a result, I am sticking with the Classic version at least for whatever time I can still get files converted.

I am also experimenting with a Perl script that will convert a series of selective .PDF prints of the board design into the standard set of Gerber files. It is claimed that this method works for both the Classic ExpressPCB designs and the newer ExpressPCB Plus designs, which makes perfect sense as the script is converting .PDF files, *not* the actual design files. The trick is in getting the correct aspects of the design printed successfully into the .PDF files that the script will read. Of course, a Perl environment will also be required in which to run the script. I use Strawberry Perl for Windows right now, though I am only beginning my experimentation with this process. I may write another short article about the process once I have it working satisfactorily.

I believe that the ExpressPCB board cost is way out of line compared to the prices that I pay, and have paid, elsewhere. A 2" x 3" 2-sided PCB set ordered from JLCPCB starts at just $2 USD plus shipping (about $18 USD for DHL Express) as compared to about $71 USD plus shipping for the same board set ordered from ExpressPCB, with another difference. The $2 JLCPCB price is for five (5) copies of a board, while the $71 ExpressPCB price is for only three (3) copies of the same board. Most hobbyists cannot afford to pay the ExpressPCB prices for their boards, especially since in most cases, only one copy of the board is actually needed.

The Classic ExpressPCB v7.9 .PCB files that I get converted to Gerber file sets are converted by a company by the name of Stewart Instruments for an $11 (USD) fee for a double-sided board. Go to their web page (http://stewartinstrumentsinc.com/) set up for this purpose and complete the form found there. Then simply follow the instructions on the form to submit your

.PCB file for conversion. Shortly thereafter, usually within thirty minutes, you will receive an email with the Gerber file set attached.

To order the PCB's from JLCPCB, navigate to their website (www.jlcpcb.com) and sign in as a user. I sign in with my Google account, which works well. Then, upload the Gerber file set as a .ZIP or .RAR file, fill out the order form on the landing page, and save it to your cart. When you have ordered all of the boards that you want in that order, go to your cart and check out, paying for the job. It usually takes three days for the boards to be completed, and then up to another possible week for the package to arrive via DHL Express, depending upon Customs activity. The boards are shipped in a carton, shrink-sealed in heavy plastic with a silica gel packet inside to protect the boards from moisture.

I have never had any problems at all with this system, and I have ordered a total of forty-two different board designs through them to date, plus some re-orders of certain boards, bringing the total order count closer to fifty orders, all trouble free. The most I have ever paid for a board order there is $11.80 plus shipping, for some fairly large boards. Price is determined by the size and type of the board, not by its complexity. They are a great company with which to do business. An example of their board quality is shown in Figure 13. This board was designed by me in ExpressPCB and produced by JLCPCB.
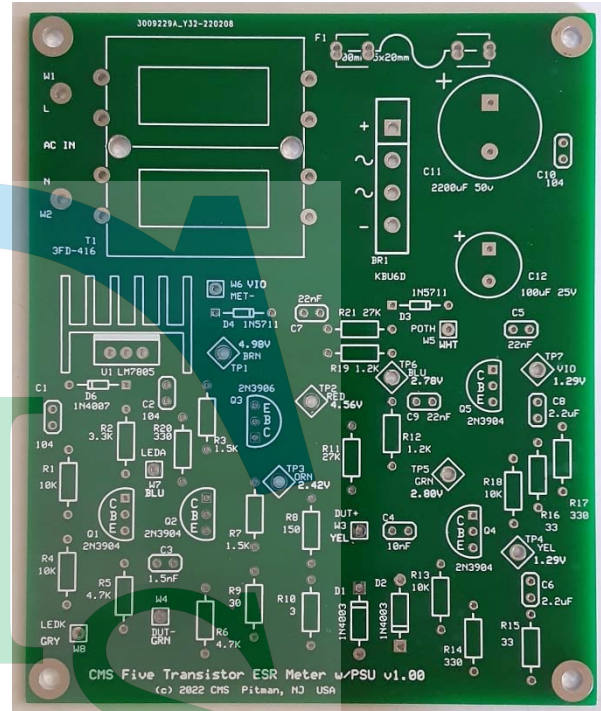

Figure 13 - PCB from JLCPCB

So, what is the bottom line here? The bottom line is that no matter how you choose to produce your printed circuit boards, do not be afraid to try. It is extremely rewarding when you have a professional-looking PCB at the heart of your project. It goes a long way to lending credence to your status as a serious electronics hobbyist or technician rather than just being a dabbler. Sure, there will be some mis-steps along the way. For example, I once, early on in my design days of having boards made outside, designed a board of which I was especially proud, as it was the first time that I used a ground plane and thermals for ground connections to that plane. I took great care to ensure that no ground points were isolated in islands, even adding vias and top-side traces to join segments of the ground plane as necessary. Thus, I was understandably upset when I assembled the project and found that my five-volt supply did not work, because the +5V voltage regulator did not work. I finally tracked the problem down to a minor but important problem with my board design, in that I had forgotten to connect the ground pin of the LM7805 +5V voltage regulator to the ground plane. In production, the problem was solved by laying a jumper wire from that pin pad on the board to a grounded point and soldering it in place. The point here is that you *will* make mistakes. Do not let them discourage you.

Check your schematics carefully before going into board design. I never go from schematic design directly to board design. Instead, I wait at least overnight for the circuit design to

percolate in my brain, going over each aspect of the design for correctness before committing it to a PCB. I do the same thing with my PCB designs. I never rush them to production now. Instead, I again wait at least overnight, allowing the PCB design to penetrate my brain, checking it for errors before committing it to production. Even then, errors will happen, even when there is nothing wrong!

I recently built a power supply having both positive and negative 12VDC outputs. The circuit design went well, as did the PCB design. By the time that the PCB's came in, I had already completed the entire remainder of the power supply assembly. All that was left to do was to stuff the PCB, solder everything in place, and install it into the enclosure. I did all that quite happily, and then powered the supply up for the first time. As it turned out, the -12VDC output was right on target, but the +12VDC output was not. I started investigating the problem, and it did not take me too long to notice that I had installed the +12-volt voltage regulator backwards, despite the clear markings on the PCB. A simple fix – desolder the v/reg, reverse it, and reinstall it. Problem solved. Again, my point is that even the best technicians will occasionally make mistakes, and I am not in any way among the best technicians out there. Therefore, I should make more than the occasional mistake. This was one of them.