Tech corner – Your own PCBs, Part 2

In the previous article, I discussed setting up ***ExpressPCB Plus*** and the schematic capture of a two-channel electret microphone mixer with an adjustable band-pass filter.

To minimize cost, ***ExpressPCB Plus*** limits you to one board size, 3.8" x 2.5", with two layers. You can waste space, you can put multiple copies of your circuit on the board, but if you want something bigger with more layers. you'll have to pay extra. If you're really tight on money and the circuit is simple, you can order boards without a solder mask and silk screen.

Abbreviations

We're going to work with modern components (no vacuum tubes), and minimal through-hole parts. These abbreviations might be useful when ordering online, so it helps to speak the lingo:

DIP (dual-inline package) : a type of integrated circuit package with pins for connections; can be placed in a socket if needed

EOL (end-of-life) : an obsolete part or one that didn't sell well; order a bunch if you can, but look for a replacement

footprint : appearance of an electronic part layout on the PCB, including vias, pads, silkscreen, copper

haywire : wire, typically insulated #30 used to fix mistakes

QFN (quad flat, no-lead) : QFP but without leads; almost impossible to solder without heat gun or reflow machine; avoid if possible

QFP (quad flat pack) : integrated circuit, like an SOP/SSOP/MSSOP, but with leads on all four sides

silkscreen : white paint used for part numbers, outlines, and other notes

SMD (surface-mount device) : wire lead-less component that's soldered directly to the PCB

solder mask : (historically) green paint on top of the PCB to keep solder from bridging between adjacent traces or pads

SOIC / **SOP** / **SSOP** (small outline package) : surface-mount integrated circuit with varying spaces between thin leads

trace : copper connection between two locations

via : hole lined with copper plating through the PCB

ExpressPCB Plus

ExpressPCB Schematic is loosely coupled to the board layout program. Unless you have two large monitors, I suggest you print the schematic for reference, to avoid clicking back and forth between screens.

1. Follow the help; there are tutorials and sample boards from simple to complex, to help you learn. If you already know a layout program such as ***KiCad*** or ***Altium***, just make sure you have a "handy list of expletives and know how to use them" (™*Calvin and Hobbes*).
2. Determine which parts and part types you're going to use, which you might already have included during circuit capture.
3. Make sure your parts are available. Some parts become obsolete quickly, others have six to twelve months of lead time if they're out of stock, particularly integrated circuits. Don't order boards and find that your parts are EOL or won't be available for many months.
4. The ***ExpressPCB Plus*** has a small library of built-in footprints, but these are seldom enough. Learn how to make your own library; you'll accumulate quite a few over time.

Parts and cost

Your funds are most likely limited. The bottom line is that SMDs are cheaper than through-hole. For example, ten 1-kΩ, 5% 1/8 watt through-hole resistors will set you back 43¢, while the 0805 SMD equivalent costs 38¢ for ten. Once you get the hang of it, SMDs

are quicker to solder than through hole: you don't have to flip the board over, applying flux with a pen is easier, and you don't have bits of wire leads flying about the place.

A downside to SMDs, especially for small ones, is a lack of marking. You can usually tell the difference between a capacitor and a resistor but you'll probably need a **multi-meter** to figure out what value it is.

The final problem is sneezing and too much coffee. Once an SMD resistor falls on the carpet, it's gone. SMD parts come on reels for the automated "pick-and-place" machines used to assemble most consumer electronics. Small parts will be packaged in stamped, thin cardboard with an adhesive plastic cover. When they arrive, label them with their values and sizes. You may wish to cut them every 20 or so for storage. Make sure all are labeled, or you'll end up throwing them away. Instead of cardboard, larger parts will have a plastic backing or come in a tray.

Figure 1: 0603 ceramic capacitors, 1411 tantalum capacitors, FT232 SOP ICs, penny for size

Connectors

PCB connectors can easily become the most expensive part of any project. The easiest to work with are 0.1" headers, male and female. These will be found on Raspberry Pi and Arduino boards; you can design your PCB as a "hat" that connects to the underlying board. You can even stack these to build up some significant circuitry. Other common connectors are 3.5 mm DC power, USB type B and USB-C (useful for power), old school DE-9s, and RJ-11/RJ-45 jacks. For antennas you can get SMA and BNC jacks upright and 90 degree for PCBs. As I said, it's easy to spend more on the connectors than on the rest of your project.

Figure 2 : Common PCB connectors

Ordering the Parts

There are three main parts suppliers for hobbyists: **Mouser Electronics** (https://www.mouser.com/), and **Digikey** (http://www.digikey.com/), and **Jameco Electronics** (https://www.jameco.com/). Their online cata-

Tech corner – Your own PCBs, cont'd

logs are great places to check on parts availability, and all three provide excellent service. When the parts arrive, put them in a box labeled for your project especially if you have several projects running at once. But if you can't wait for a part, your local electronics store may have what you want: connectors, battery holders, wire, through-hole parts, NTE ICs, and a myriad of other things you won't need until you do.

Creating a new board

If you've followed these instructions and captured the circuit, you're ready to create the PCB.

1. Fire up ***ExpressPCB Plus***.
2. From the ***Gallery***, select ***New*** and fill in the required board information.
3. You should get a blank 3.8" x 2.5" area sized to fit the screen. ***Red*** front side copper, ***Green*** back side copper, ***Yellow*** front side silkscreen.
4. Place your most important components. Life is easier if these loosely follow your schematic diagram. Lay out all the components before you begin adding traces.
5. Place the smaller passive components nearest to where they connect to things, the shorter your traces, the happier you and your circuit will be.
6. Place mounting holes on the corners with sizes suitable for the screws you'll use. #4 screws work nicely with a 0.125" hole, plated or not.

Wiring it up

Start with the ground connections. Typically, the back of your PCB will be nearly completely covered with copper that's connected to signal ground. For some simple parts, you merely need to place a plated via to the back side. You still need to tell the system what to do on the other side. For complex parts, follow these steps:

1. Select the part, right click on it.
2. Click ***Ungroup*** on the menu that pops up.
3. Select the pin/via to ground, right click on it.
4. Click either ***Bottom Copper Thermal Isolation*** if the via is part of a pad, or ***Bottom Copper Solid*** if it's just at the end of a trace.
5. Click the ***X*** to kill the menu, the change has occurred.
6. Select the area of the part by left click and drag.
7. Right-click the area and select ***Group*** from the popup menu.

To run a trace, click the ***Trace*** button and start left clicking along the path. Right click to finish. There's no auto-router.

Ground plane pour

Once you're happy, and everything's labeled, it's time to pour the ground plane:

1. Select ***Rectangle***.
2. Click the ***Pour*** symbol (looks sort of like a paint bucket on top).
3. Select ***Bottom Copper***.
4. Make the rectangle cover the area to pour.
5. Final check: are there isolated ground areas? That is, an area with copper that's not connected to the rest of the plane. You might need to move some vias or traces around so they're all connected, particularly if there's a ground via from the top.

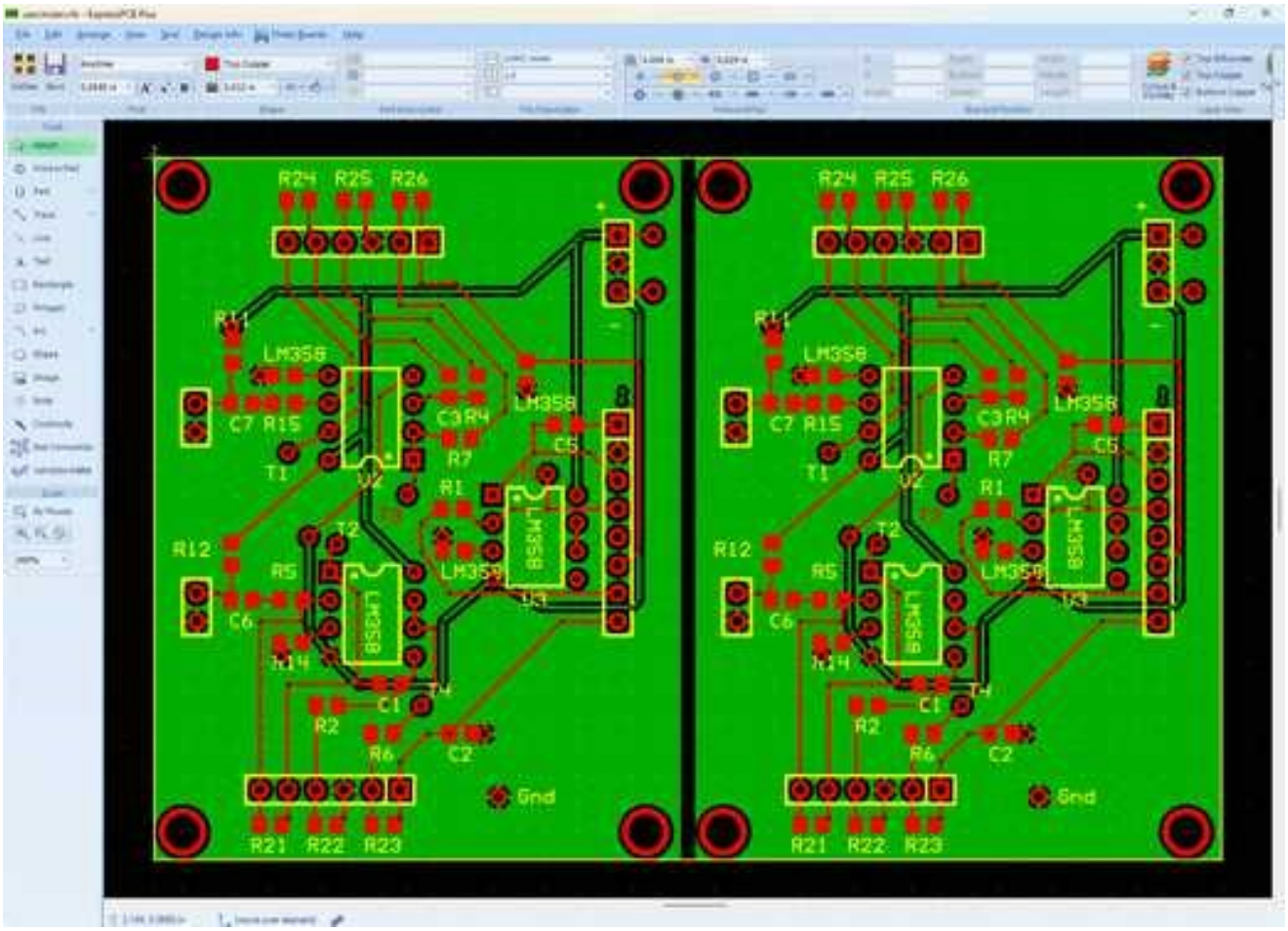
Final checklist

- Are all your ground vias actually connected to the ground plane?
- Are all your silkscreen annotations actually silkscreen and not copper?
- Is your ground plane totally connected? If there are areas not connected to the rest of the ground plane you may have to move traces so the fill algorithm can connect other areas to this one. If

not, you might need to add an "above ground" trace to hop over the congested area.

- Are some of the traces too close to vias or other traces?
- Have you correctly established which is pin #1 on your integrated circuits?
- Are all pins hooked up that need to be hooked up?
- Have you annotated the version of your board in some way?
- Do you have the right footprints for all parts? A 603 resistor can be soldered onto an 805 pad set but vice versa not so good.
- Do ground vias that will have solder on top have thermal isolation on the bottom copper?
- Do you have mounting holes for screws?
- Is there sufficient space around ICs for you to use a socket? Most parts don't have an outline and IC sockets are space hogs.

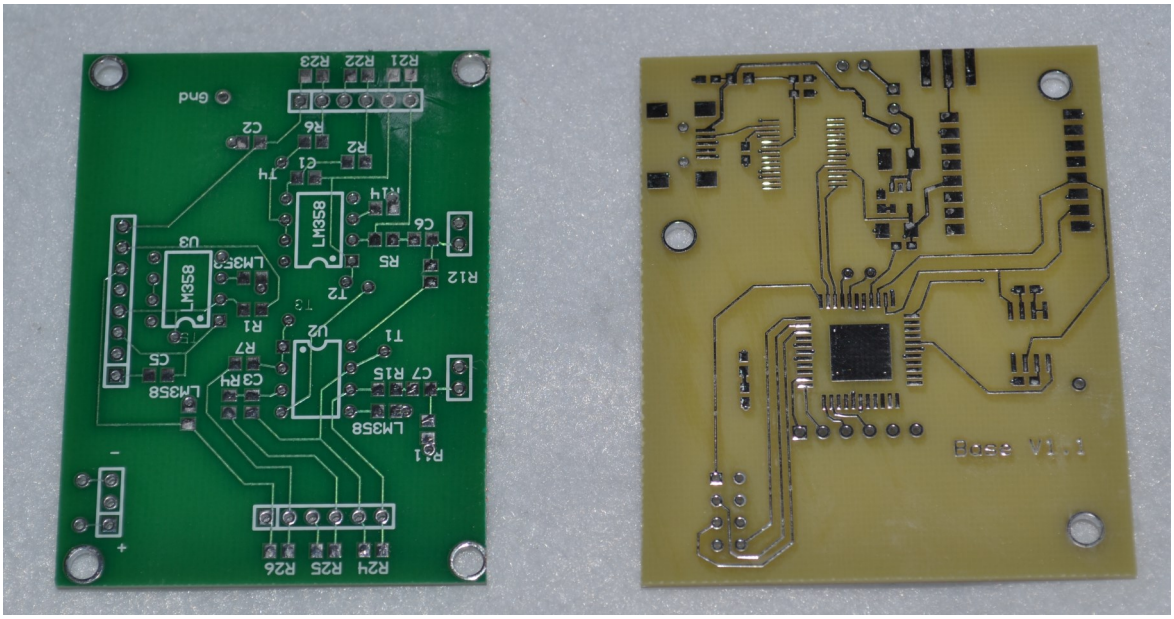
After completing the circuit on the left hand side, I grouped and copied it to the right hand side. The results looked like this:



Tech corner – Your own PCBs, cont'd

Ordering the boards

Click on ***Order boards*** and decide what services you want. I opted for the *solder mask* and *top silkscreen* which costs extra but makes your life much easier. Give 'em your credit card, and let them do the work. Less than 2 weeks later, here they are.



Left: mixer board with solder mask. Right: bare LoRa board

Next month

Assembly, box printing, testing, hay-wiring.

Jed Marti KI7NNP