Intro to Printed Circuit Board (PCB) Design

Eric Ayars (Cal State), Gabe Spalding (Illinois Wesleyan)

December 20, 2022

Motivation

Previously, you've put together your circuits on a breadboard. Why should you bother with making a **Printed Circuit Board** (PCB)? Whenever you want to make multiple copies of your circuit, moving to a PCB-based design very quickly becomes worth your while. A printed circuit board can also be a better option when it comes to dealing with high-frequency signals or when measuring phenomena occurring over very short timescales. Another factor might be that breadboard-based designs tend to suffer from crosstalk and significant noise pickup. Overall, surface-mount designs *perform better* than circuits with through-hole components, and can be cheaper!

The goal for this worksheet is to cover the most useful first stages of the PCB design learning curve. We're not going to cover *everything*, but will instead emphasize that there's really just a small subset of the options available that you will need in order to draw up a useful circuit schematic and then turn that into a board layout design that you can submit for fabrication.

There are many free PCB design programs out there. KiCAD has the advantage that many manufacturers (and distributers such as Digi-Key, Mouser, and Jameco) specify the "footprint" of their components as KiCAD libraries, but KiCAD itself has long been little more than a collection of programs held together by a mostly-common interface. FLUX.ai is a newer effort that aims to incorporate KiCAD libraries into a more coherent software package, which we intend to explore in the future. For now, we'll discuss KiCAD.

Online Resources

- KiCAD: Download and install KiCAD 6 onto your computer.

- Simple supplement: Build Electronic Circuits site
- Full tutorial: Getting Started in KiCAD

Your first printed circuit

We're going to keep it simple by starting with an op-amp based non-inverting amplifier. As is traditional for any electronic design, we start with a sketch on the back of a napkin. (See figure 1.)

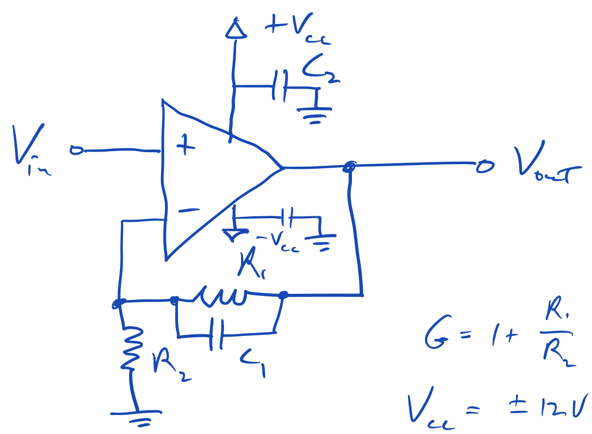

Figure 1: Typical napkin-scrawl. (Spilled beer omitted for clarity.)

We want to change this rough sketch into a testable schematic, and from there into a board design that can be fabricated and built into a working circuit. Accordingly, the two main KiCAD programs we will use here are the *Schematic Editor* and the *PCB Editor*. These both do just what their titles indicate: the first one lets you build a schematic, and the second one lets you turn that schematic into a board design.

Schematic step-by-step instructions

1. If you have not done so already, download and install and run KiCAD.
2. Create a new project, and click the 'Schematic Editor' icon. If working correctly, a blank template will appear. It is well worth your while to

go to ‘Page Settings’ under the File menu to set the paper size, author, title, version number, *etc.*

3. Now let’s add parts. Third-down on the right hand side there’s an icon that looks like an amplifier. Click it (or press ‘a’) to select the ‘add parts’ tool, then click on the schematic. After a moment or two, approximately 17,000 options will appear (not an exaggeration). Fortunately, there’s a filter available: type in ‘TL061’ for a TL061 op-amp, and place it on the schematic. Continue this process for the resistors and capacitors, placing them roughly where they’ll be needed. Note that there are options for which symbol to use: if you filter by ‘resistor’ you will see many options including ‘R’ and ‘R_US’. The symbol you are most familiar with is ‘R_US’. (For the sake of anyone else who has to look at your schematic, consistency is recommended!)

4. As you are placing the parts, note that pressing ‘r’ rotates the part $90°$ counterclockwise. You can also rotate parts later by right-clicking on them and selecting from the menu of options given.

5. This board will need connections for your input and output signals. For high-frequency signals, these are best provided by BNC or other co-axial connectors. You may want to mirror one of the connectors, which you can do by right-clicking on the connector after it’s placed.

6. We also need three power connections: $+V$, $-V$, and GND. These can be physically provided by screw terminals. You should also place voltage symbols, such as $+12V$ and GND, with the power placement tool just below the ‘add parts’ tool on the right. One convenient trick is to place multiple symbols: you can for example put as many GND symbols on the schematic as you like, and they’ll all connect to the same network. I make use of this trick extensively on my designs: I’ll have $+12V$ symbols on the op-amp and then have the screw-terminal block with $+12V$ somewhere in the corner of the schematic.

7. By this time you probably have a bunch of parts scattered onto the schematic and it’s a mess. That’s fixable. If you use the selection tool (the arrow at top of the right of the window) and click on any part you can Move it by pressing ‘M’. You can Rotate it also (‘R’).

8. For adding wires, the tool is the 5th one down on the right, it looks like a diagonal black line. You can also select this tool by pressing ‘w’ for Wire, or by clicking on the terminals at the ends of each part.

Now, wire up your circuit. Move/Rotate parts so that the schematic is 'pretty', and make all the connections. As you are doing this, it may be helpful to draG ('g') components instead of Move ('m') them. If you draG them, the wires stay attached! (This is sometimes a convenient way of checking whether the wire is attached.)

9. There are things to be very careful of when doing the wiring! Be sure that the wires go to the *terminals* at the ends of the parts; otherwise they're not really connected. Also understand that crossing wires don't connect by default. In figure 2, $-12V$ is *not* connected to GND! If you want two wires to connect, use the Junction ('J') tool and click on the intersection.

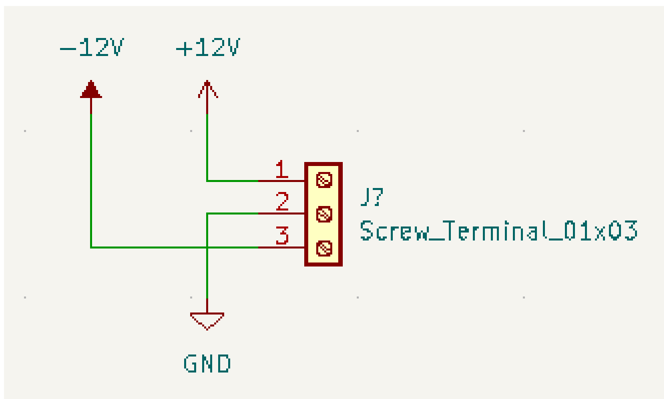

Figure 2: Screw-terminal power block segment of the schematic. The negative supply is *not* connected to GND.

10. KiCAD's error-checker routines need to know where power is coming in. To tell the program this, it's necessary to attach a 'Power Flag' to the $\pm 12V$ and GND symbols at some location in the schematic. These are most easily found with the power placement tool. (I connect them at the power/screw-terminal part of the diagram.)

11. It's nice to have component values on the schematic. right-click each resistor and capacitor and enter the value ('v') for that component. I used $r_1 = 100k\Omega$ and $r_2 = 10k\Omega$.[1]

[1] These particular values were chosen because we have an overdeveloped sense of humor.

12. One more thing: The TL061 chip has two pins that we're not using in the napkin-based design shown in figure 1. Those pins are for the offset adjust. If you would like an offset adjust trimpot in your circuit, go ahead and connect one. Otherwise, we need to tell KiCAD that those pins were not forgotten but were deliberately left disconnected. The eighth icon down on the right lets you place 'no-connect flags' on the circuit. Use this tool to put $\times$'s on the unused pins.

13. If everything looks good, now it's time to fill in what KiCAD refers to as the references. So far each part is '?': R?, C?, *etc.* Rather than change these all manually (and risking having two parts with the same label) have KiCAD do it for you. Under the 'Tools' menu, click 'Annotate Schematic'.

14. All done with the schematic? Any errors? We can check! Under the 'Inspect' menu, choose 'Electrical Rules Checker'. Fix any errors it flags, and you're ready to start the second half of the process!

Board Layout step-by-step instructions

Now that we have a schematic, let's turn that schematic into a board. First, we have to tell KiCAD the 'footprint' for each piece. This is one of the trickier parts of the process, and doing it right is key to success. KiCAD then generates a 'netlist,' a list of part shapes and description of which pins are connected to which other pins. We then arrange the parts on the board and route the wires.

1. Under the 'tools' menu, select 'Assign footprints'. KiCAD opens another window with three columns: Footprint Libraries, Footprint Assignments, and Filtered Footprints.

2. The Assignment column will be alphabetical, so start with C1. (For me this was a 1μF bandwidth-limiting capacitor in the feedback network on the op-amp.) We are introducing Surface-Mount Devices (SMD), so in the Libraries column select the 'Capacitor_SMD' library. The right-hand column will now show all the footprints in that particular library category. I cannot overemphasize this: *The footprint you select must match the part you intend to use.* SMD parts come in a very wide range of sizes and packages. Some of these are too small for human placement! I personally am comfortable working with 0805-sized parts, and although I have successfully hand-placed 0603 parts

when necessary, I recommend selecting 0805 or larger. (The parts we'll be using in our 'build' worksheet are 0805 or 1206.) In addition, there is the option of regular or 'handsolder' footprints. The handsolder pads are slightly larger and easier to work with. Double-click the desired footprint to assign it to selected part.

3. Assign the rest of the parts. The most common BNC connector is the TEConnectivity_1478035_Horizontal, the TL061 is a SO-8 3.9x4.9mm, and the screw terminal fits a 4Ucon horizontal with 3.5mm pitch.

4. So far, the op-amp specified in your design has been a TL061, but let's suppose that you were making a different circuit that required a higher-bandwidth op amp, such as an LF411. Unfortunately, the default KiCAD libraries do not include the component footprint for this part. While it is possible for you to design your own, it is much simpler to go online and download it from SnapEDA, which does require you to register as a member of the community, but remains free so long as you are downloading fewer than half a dozen footprints each week. Even though it might, at first, look as though their version of the LF411 is not a surface-mount component, that package choice really is included in what you'll download. [Again, distributers, such as SparkFun, all provide links allowing you to download KiCAD component footprints for everything they sell.] Once you've downloaded a library and model, you can simply add that to the working folder of your project (so that anyone you give that to will have all that they need). If you want to add module for all of your future projects, select KiCAD Prefs ≫ Manage Symbol Libraries, and add what you wish.

5. Once all component footprints have been assigned, click Assign, and then click OK.

6. Now it's time to go lay out the board. Click the 'Open PCB in Board Editor' icon at the top right. You will again get an engineering diagram, but this time with a dark background. Set up the page details with File/Page Settings.

7. Press F8 (or click 'Update PCB from Changes Made to Schematic') to have all your parts magically appear in a useless cluster with a spiderweb of lines (usually referred to as a *rats' nest*) showing what should connect to what. This is your starting point for the circuit layout, and it will not even be close to how things should be.

8. Depending on your project, you may have a size constraint for the board. In that case, you would draw the board outline now. (See step 15.) In this case, though, we're going to build the circuit first and then see what the board should be shaped like. (That's often easier.) But start thinking of the small middle area of this sheet as the board, and put your parts on that board using Move and Rotate ('m', 'r') to place them in a way that will minimize the tangle of circuit traces needed.

9. As you place parts you will notice that they snap to a grid. The default grid size may be too large, feel free to change it. Also, be careful when moving things: it's possible to move parts of the footprint label without moving the footprint, *etc.* Effective practice is to select one of the pads and then press 'm' to move the entire part.

10. Once you have a reasonable part layout, start running wires between pads to replace rats' nest wires. Use the 'Route Tracks' tool, *not* the 'Draw Lines' tool. The board has two sides, front and back. With through-hole electronics, the back side is preferred. With surface-mount, though, the parts are on the front side and front-side traces are preferred. Start with the short/obvious/easy wires on the front side, but put off all of the GND connections (as we'll do those last).

    Effective Practice: The traces are made by etching away copper on the board. If you have acute inside angles, etchant residue tends to stay there and gradually eat at the board, potentially causing an eventual failure. For this reason, avoid having traces meet at acute angles, and two 45° angles are better than one 90° angle.

11. Once the easy connections are made, start filling in the longer ones. (Again, don't do GND yet; we'll do that last.) You will eventually find a trace that you can't efficiently route on the front side without traces crossing. To solve this, you'll have to place a via to take the signal to the other side of the board. Route the wire on one side until you get to where you want to tunnel through the board. Click to end the wire there, then press 'v' to place a via. Select the opposite layer at the top of the window, and continue routing the wire on the new side as needed.

    Effective Practice: With wires going from a through-hole part to a SMD part, it's a good idea to start the wire on the back side at the through-hole part, then bring it to the front to attach to the SMD part. The board-fabrication houses I recommend all use plated holes, so this is not technically necessary, but it's probably a good idea just in case the board-fab house does not plate holes by default and you forget to tell them to do so.

12. For the last set of connections, let's make a ground plane. Select the 'Add a filled zone' tool (right side, 7^{th} icon from the top) and use it to draw a rectangle matching the border of the circuitboard. When the pop-up window asks, tell it that the filled zone will be 'GND', on both top and bottom Cu layers. Once completed, this filled ground plane should connect the last of your rats' nest wires and the 'unrouted' count at the bottom of the window should go to zero.

13. It's not zero, is it? ...Zoom in, find the unconnected wire, connect it. Sometimes there's just a small gap between the end of the trace and where KiCAD *thinks* the trace should end. In extreme cases, I've gone so far as to turn off all layers other than the rats' nest, just to see where the last problem was.

14. I have a touch of CDO[2] so I tend to spend *way* too much time on this, but go through and clean up the design a bit. Are your parts aligned so the BNCs are an inch apart and extend the same distance? Would switching a pair of components, or rotating one, allow for shorter/cleaner traces? Do you really *need* so many vias? If you rip up and/or reroute any wires, the ground plane you made won't look right. Pressing 'b' redraws it.

15. If not done already, draw the edge of the board. Change the layer to 'Edge cuts' and draw a border using lines (not wires).

16. As long as you're drawing the edge, put mounting holes in also. Click the 'Add a footprint' tool, search for 'hole', and select the size of hole you want to use. (M3 is standard, and works well with either M3 screws or 4-40 if you've not switched yet.) You can't delete the (useless) text associated with these mounting holes, so we recommend simply moving that to a location outside of the board area.

17. Now it's time to do the silkscreen. This does not affect the functionality of the circuit, but it can make it easier for the user! Deselect most of the layers, other than F.Cu, F.Silkscreen, F.Courtyard, and Edge.cuts. Toggle layer F.Fab and decide whether any of the text in

[2]OCD, but properly alphabetized.

that layer is something you want to have on the silkscreen layer. If so, select the text and right-click on it. Select ‘Properties’ (or just press ‘e’) and change the layer to F.Silkscreen. Move and/or rotate component labels so that they’re all in reasonable locations.

18. Label the connections. $-V$, GND, $+V$, Input, Output. Use the ‘Text’ tool. Make sure the labels are on the F.Silkscreen layer!

19. Add your name and circuit information to the silkscreen. You can do this on either the front or back: front is more visible but there’s more room on the back. Use the ‘Text’ tool. If you put it on the back silkscreen, make sure it’s reversed!

Generating Gerbers

I’ve given up on fabricating my own boards. Yes, I can do it and get reasonably good results, but fabrication houses are cheap and fast so I send them out. It takes a week or less to get perfect boards with lovely silkscreens and soldermasks and working vias. All fabrication houses use Gerber files to make the boards. To generate your Gerber files,

- Select ‘Plot’ under the File menu. (Not ‘Print’.)
- The layers you should include (Front and back) are Cu, Paste, Silkscreen, Mask, and Edge.Cuts.
- The rest of the defaults are usually right. Click ‘Plot’.
- Before you leave that page, also click the button to generate drill files. The default format, Excellon, is the one you want.

Once that’s done, you’ll have an extra half-dozen .gbr and .drl files in your working directory. Go back to the main KiCAD control program and select the Gerber Viewer program. Open the Gerber files and make sure each layer looks like what you want. Before spending money to have these fabricated, you might print them out on paper, using a standard laser printer, just to check that your components actually fit the footprints you’ve specified.

If you’re satisfied, and certain it’s right, gather all those .gbr files and the .drl files into one .zip file and you’re ready to send them to fabrication. If you plan on making twenty boards, we highly recommend that your order should also include a stencil, which is used for quickly applying solder paste, to all pads, on board after board, very quickly!

The fab houses I use are

- AllPCB.com. This place is fast and cheap. I believe it's located in Guangzhou China, and my children assure me that they're manufactured by child slave labor but I've not been able to verify that. It typically costs less than a dollar per board, and about \$30 shipping, and I get the boards in 5-8 days.

- Osh Park. These people are an aggregator: they collect a bunch of orders, put the orders on one big circuitboard, and then have someone else (child slave labor?) manufacture them and cut them apart. Fabrication time can vary, depending on order volume, but board quality is quite high and the boards are purple. They are a US company, if that is of importance to you. Their cost is \$5 per square inch, and boards are ordered in multiples of 3. Submission is as easy as dragging a zip file onto their web page.

There are other options also, of course: Google it if you don't like either of those. (If you find a US-based fab house for a reasonable price, let me know.)

Submit your .zip file through the website of your chosen fab house, and wait for a week.... Once you have a printed board in hand, you're ready to move on to our next worksheet, "How to attach Surface-Mount components to a Printed Circuit Board (PCB)."