A New Hobbyists Perspective On Learning Electrical Engineering

Peter Rieke, March 22, 2025

Over the years, I have kept an eye on electrical engineering design tools and have always found the professional ECAD versions to be too expensive and the semi-hobbyist versions like Eagle to be very clunky and difficult to use. Further it was difficult to have Printed Circuit Boards (PCB) fabricated at reasonable prices and too complex or dicey for home fabrication (even for a chemist). But finally, we now have open-source ECAD software and fabrication houses that make designing and building circuits relatively easy and hence of minimal monetary risk. Heck, give it a try and make a simple board and give them away as Christmas presents. It's that cheap.

The key elements of creating a working electrical circuit on a PCB are:

1) Creating a schematic
2) Laying out the PCB board
3) Finding specific part numbers from supply houses like Digikey and Mouser
4) Ensuring the footprints of the part match that of the PCB layout.
5) Sending off all the data files to a PCB fabricator
6) Purchasing the components
7) Soldering all the components on the PCB
8) Testing

KiCad

KiCad cracked open ECAD and provides an almost professional schematic and PCB layout tool. It is open source and it costs nothing to download the full feature version -- although one could make a donation if you like the product.

The schematics are easy to create and modify and with minimal learning curve one can figure out how to insert a symbol and associate it with a footprint. But often one needs a custom symbol or footprint not in the initial libraries. In my opinion the symbol, footprint and 3D model libraries of KiCad are cumbersome. It has not been clear how and where to download these files and then activate them for use in a project. Some trial and error will be required. I will write more on my perspective of the workflow in KiCad. Workflow, for those unfamiliar with the term, is just the order how things should be done as well as iterative aspects of the process.

Layouts (the PCB design) is also relatively easy. The major problem is not the software but in learning how to organize components. One has to be willing to try a design and then throw it out and start over. I start by laying out the schematic so I have a clear understanding of signal and power flow in the circuit. This helps me arrange parts into subsections. Fortunately, if one selects a set of components in the schematic these are also selected in the PCB layout and can be moved as a group.

Then one generates the so-called Gerber and Drill files and sends these off to a production house.

Production Houses

I use JLCPCB. They are a Chinese fabricator that is one of the countries "loss leader industries" designed to produce world dominating industrial automation in electrical engineering. US fabricators are more expensive but as I would argue that fabrication is not a major project cost factor except for all but the poorest of hobbyists. JLCPCBs interface is easy to use and tailored to the hobbyist but should work for the small commercial projects too. Lots of definitions and design criteria are easy to look up if needed. If you omit critical files or screw them up, the JLCPCB system will scan your files and tell you that you're an idiot but that they still like you. For simple boards shipping is more expensive that fabrication. The "slow boat from China" option takes two weeks and I, frankly, will wet my pants waiting that long so I opt for next least expensive and sufficiently fast shipping option

There are plenty of other fabrication houses out there and a US fabricator while more expensive is a good option as you move beyond the newbie phase. I'm just getting out of the newbie phase myself and that has taken about 2 years.

Parts Distributors

I use Mouser and Digikey. I prefer Mouser because the search menu is most effective. But Digikey has a component list, with part numbers, that can be directly downloaded to KiCad. Regardless there is a learning curve to finding parts efficiently. You will read and/or scan hundreds of datasheets initially. That's why many forum commentators, quite condescendingly, will tell you to RTFD.

The datasheet is right there and one can usually find available info on available footprints. For some parts you may have to go to UltraLibrarian to get the correct footprint as well as a symbol and 3D model. There are other footprint providers but UltraLibrarian is my weakly preferred option. It is important at this point to transfer the datasheet link, the exact part number and the footprint to the KiCad "Symbol Fields Table." From this you can generate excel and/or .cvs files to upload to Mouser and generate a shopping list. This list will always be incomplete and have errors but it saves time.

Do not shy away from Surface Mounted Devices (SMD). Resistors and capacitors in the 1208 and 1210 sizes are pretty easy to solder with a bit of practice. I can do 0603 caps but it is not fun or quick.

My Electronics Lab

When I decided my next biggest project was going to be the design, fabrication and testing of an inexpensive, open source, water quality instrument, I needed a lab. I had my carpenter guy build me new benches that were ideal for wheelchair height and populated with lots of power on two existing circuit breakers. One half was for chemistry while a larger L-shaped section was committed to electronic fabrication and testing.

Computer Tools

On my L shaped bench, the left leg is dedicated to a computer display hooked up to a cheap Lenovo laptop. This is mostly used for referencing KiCad during builds, keeping assembly notes and transferring files between my other computers. It also runs the Arduino IDE for writing and downloading code to Arduino Boards. I use a USB splitter to get more ports and adapt to the different kinds of USB ports.

Arduino

The USB splitter connects to Arduino MEGA 2560 modules because, although a bit long in the tooth, these run just fine for what I want to do. They have plenty of analog, digital and communication configurations. I can write Arduino scripts or code in C. I have mounted the modules along, with two pieces of breadboard for prototyping circuit concepts, on some pressboard with rubber feet. I mostly interface with power and an Analog to Digital Converter (ADC) as well as a second USART serial connection to the computer. So, I don't need much and could use a much smaller processor for most applications. But the software won't change much and, for development, more is better. As I am working with a power supply for analog circuits (+12V, -12V), the power supply also can supply +5V for the Arduinos. It is best to not provide board power through the main Arduino USB port and hence a second USART serial bus without power is useful. I use the Arduino IDE through the main USB port to debug programs but use the second USART and Putty to transfer data when the analog power supply is being used.

Testing

My test and power supply modules fit in the corner on a small rack. I have the low-end equipment for serious hobbyist use. I have an oscilloscope, a 5-amp DC power supply and a signal generator. All of these are the Rigol brand equipment. Seems to hold up and gives results that are good down to about 2 mV before noise creeps in. These are not precision measurements. I also have a Fluke 173(?) multimeter. Between these items, the computer and screen, and the Arduino boards I have about $3K blown to bits.

I remember when an analog oscilloscope was huge and expensive and I was hesitant to buy one for that very generational misconception. But the cheap ones for few hundred bucks are worth the money. In dealing with analog circuits, you usually are looking for an AC or DC signal but noise and interference from other devices is problematic. The scope is invaluable in identifying the frequency of signal response problems. In some cases, I can detect the LED light hanging over my main work area.

Assembly

Coming around to the right is my area for fabrication and testing. Things get messy during assembly and I use a silicon sheet to work on (I should use a grounded sheet but this is what I had). Once I clean up from assembly, I can shake the sheet off and have a clean surface for testing. To my far right, I keep three soldering stations. One is an old Radio Shack adjustable station with 3 set points. It was built by another company but was rebranded. It works well and changing temperatures is a single push of a button. The next is the common Hakko model FX-888. It is adjustable and has 5 setpoints but the button punching sequence is not immediately obvious and really a bit cumbersome to switch between temperatures. I like to drop my irons down to 150C when I'm fooling around with prep stuff. Then I have to punch up "n" times to the desired temperature and then back down to 150C. The third is a X-Tronic 6020 hot air gun. This is very handy at removing chips from boards as you try to debug your shitty design. I did a lot of swapping of standard 8 pin, SOIC op-amps and the hot air gun made this a breeze. Not so great for resoldering and I still prefer an iron for this.

I also keep a rack of both solid and multi-core wire as well as bare copper wire. Most in #22 gage and a variety of colors. The solid wire is used for breadboarding and jumpers between circuit subcomponents, while the multi-core wire is used for connectors. More on this later.

You will also need a spool of ~1mm diameter solder, some paste flux for big dirty jobs, some regular liquid flux for clean boards and first assembly, some copper desoldering braid is very useful. Brass sponges are very nice for a quick clean just prior to actual soldering. Some of the little tins of tip cleaner are very useful in at least partially refurbishing solder tips with too much oxidation which usually results from overheating. I should make a list. I own two (LED magnifiers. One for electronics and the other at my mechanical bench. These are worth every penny and don't skimp here.

All in all, in soldering equipment, lights, solder and misc. stuff, I probably have $1500 buck tossed down the outhouse hole.

On Soldering

I have experience in Gas, Stick, MIG and TIG welding as well as Silver brazing of bicycle parts and HVAC components in addition to soldering electronic components. In all these techniques, one must learn to control the heat distribution and the metal or solder addition. Practice makes perfect as they say. Cleanliness and flux are also critical.

Flux

On new boards and new components, I use a variety of flux pens and especially like the refillable ones. The tips get beat up but that is not a problem and I use the tips to gently clean the solder pads before making a small puddle of flux to lay the component in. Sometimes I will flux the component pins on old parts or difficult to solder parts. Once all soldering is complete these fluxes can be cleaned away with isopropyl alcohol. I use 1qt freezer bags to hold the isopropyl alcohol and place the entire board inside. This provides for minimal alcohol use and easy agitation. The small disposable "acid" brushes are handy for hard deposits. Inspect for random solder beads and remnants of copper fibers from solder wicking braid. These will take you by surprise.

I use rosin flux paste, when I have to rework stuff like replacing a chip or getting rid of solder bridges on very small VSSOP devices. These often require high temperatures and the paste flux can be gobbed on and it lasts longer. This is always a bit messy and cleaning is important. Copper cleaning braid is very useful is sopping up too much solder and I dip the working length, about 1/2", in flux paste before using.

Cleanliness

With new boards and components this is usually not an issue but a wipe with alcohol and keeping your hands clean is helpful. Old boards may need a light scrub with a paper towel. You can sometimes pre-tin pads but use copper braid to remove any excess solder so the part will lie flat on the PCB.

Solder

I have an ancient spool of flux core Sn60/Pb40 solder. As solder is expensive, I use this and will probably never runout. If I were to buy new, I'd go with a non Pb solder of which the Sn96.5/Ag03/Cu0.5

composition is most common. Silver solders are a bit trickier to use than leaded solders due to the higher melting point. Get a 1mm or less diameter wire. I do not consider the leaded solder a major health hazard given the relatively small amount of time I actually am soldering. I do find the fumes from the flux irritating and have a small gentle fan in the area that blow the fumes away. A fume extractor might be better but, in reality, I do not consider the lead risk very high. The amount of lead spread around by reloading ammunition is a much greater hazard. Wash your hands before eating as lead contamination through handling of both solder or bullets is probably the most prevalent path for exposure to lead.

I built my own reflow oven and for that I use a leadless solder. More on that later.

Solder tips

I can successfully hand solder SOIC chips and resistor and caps down to 0603 footprint. For these I use a 1/16” (1.5mm) diameter bevel tip. The bevel creates a small oval that holds a drop of solder. I then press the edge down onto the pad next to the pin, pause briefly to let heat buildup and roll the solder onto the pin. Frankly my hands are not steady enough to hold 0603 devices in place long enough to get the job done quickly. So, I always start with a non-ground terminal to reduce the amount of preheating time. Do the ground pads last as the board will be preheated. Don’t design with very large thermal relief spokes around your ground pads. 0.3 mm is adequate.

I have used chisel tips for SMD parts but prefer the bevel. I do use a 2mm wide chisel for THD and put a drop of solder on one side of the chisel then touch the other side to the pad, pause and lift the chisel to deposit the solder. I often have to add a bit more from the solder roll to completely fill the hole.

I use a very fine point tip for VSSOP chips and, even then, solder bridges can occur. I have tried the drag method where you drag the solder filled tip across a series of pins. This has not worked well for me and I get solder bridges at the start of the drag and not enough at the end of the drag. Probably need to practice more.

I preset my irons to 150 C, 230 C and 250 C. Between solders when I’m cleaning up, finding parts, or otherwise fiddling around I set the iron to 150 C. This keeps the tips warm but not so hot that oxidation is a problem. For SMD and most THD devices I use 230 C. But for THD pads to grounds I may need to jump to 250 C to get enough heat.

Connectors

If anything will drive you nuts its connectors. Way too many options and connectors are often the most expensive part in the circuit. The trick is to select options that are compatible with prototyping but will allow extension to the needs of a final product. I use three types of connectors.

“Dupont” connectors – These are just the standard 0.1” pitch pins or sockets that one finds on Arduino boards and fits typical breadboards. Most hobby microprocessors like the Arduino come with sockets soldered to the PCB. One can also buy the pin version and the horizontal ones are most practical. To prepare a cable, the pin or socket metal connectors are first crimped onto a 22-gauge wire and then inserted into the plastic housing and retained by a tab. They can be disassembled with care. The drawback to these is that they don’t hold up well to constant twisting and turning. Bad connections can occur. They are also not keyed so one can reverse the polarity. On power inputs to op-amps, swapping the positive and negative polarity is a real bummer. Don't ask, don't tell, just listen as the ancients speak.

EP2.5 connectors -- TE Connectivity has the Economy Power (EP) connector system. These are relatively inexpensive and can be readily recycled from old boards with a hot air reworking station. Again, wires are crimped to pins or sockets that insert into a plastic header. These then connect to the corresponding PCB headers. They are quite durable, are keyed to prevent wrong way insertion and come with a snap lock (which can be removed as desired). The PCB headers also come with plastic pin or pins that snap into the PCB. The pitch is 2.5 mm but all the desired footprints can be found at UltraLibrarian. These are steady and reliable and bad connections are rare.

JST connectors – I have some but find the EP2.5 easier to find. I have some JST 3.96 pitch I picked up somewhere. Matt Millman has an excellent review of JST connectors. EP also comes in 3.96 pitch and I wonder if they are compatible. JSTs are apparently used in a lot of drones and model airplanes.

Crimping wires onto the metal pins and sockets requires practice. As does inserting and removing them from the plastic housing. There are tons of videos and instructions for doing this. Most systems use a crimp onto the bare wire followed by a crimp onto the wire coating. A thick wire coating can be problematic. There are tons of reviews of crimping tools. I choose a Hozan P-707 as being fairly inexpensive (<$50) and it does the job well for all crimping sizes. Needle nose pliers are helpful in adjusting wonky crimps. You could get away with using just needle nose pliers but that is usually a time-consuming, kludge job. A magnifying lens is very helpful. Practice is critical. I mostly use 2, 4 and 6 pin systems even if I have to leave a pin or two unused. This cuts down on parts and allow existing cables easy to be swapped between different projects. Again, practice crimping before trying to build a usable connector. Practice is more important than the actual tool used. The crimp pressure should be just right and my tendency is to apply too much force. Inserting the crimped wire and connector into the housing also can be tricky. Different connector systems insert differently and the release mechanisms vary. Tendency is to not get the connector fully inserted into the housing. Three arms, better eyes and tinier hands would be useful evolutionary adaptations.

Workflow

When one first starts out on KiCad the inclination is to do a generalized circuit with say a default unspecified op-amp. This gets you acquainted with basic circuit sketching techniques but is useless in determining layout of the PCB. To set up the PCB layout, one needs datasheets, part-numbers and footprints. Even just the selection of a resistor can be daunting. Through hole or SMD? Which footprint? Why are there so many different part numbers for the same item? Does the pin layout correspond to the symbol and footprint? Do I need to make a special footprint? Oh! And may God send to hell the people who design connectors.

I narrow the choices down by first going to preferred manufacturers and I ignore the cost of any component under ~$3 each. Most resistors and capacitors will cost pennies while op-amps can run a buck or two. I use Panasonic polarized SMD caps for greater than 10uf and Murata for the non-polarized caps. I use Yageo resistors with occasional exceptions. Texas Instruments and Analog Devices are the go-to providers of op-amps and other analog and digital components. I like Murata for their [SymSurfing] online tool that gives me the equivalent resistance and inductance for all their multi-layer ceramic capacitors.

I almost always go with SMD rather than through hole (THD) devices. Contrary to one's initial assumption, SMD components are easier to solder and much easier to remove and replace. THDs always clog with solder and require higher heats and more paste to remove than SMD devices. A three pin THD is a major headache to remove with a soldering iron. I always solder THDs slightly up off the board and cut the wires a little long so that I can, clip off the device and grab either the top or bottom of the wire and pull it out. I have also modified the footprints for polarized Panasonic caps to enlarge the contact pads. The extra area helps the iron tip wick in solder without touching the cap and have it jump out of place.

Soldering Ovens

Build one, do it as soon as you get everything else in place. I bought a cheap $50 toaster oven, a $30 dollar temperature ramp controller on eBay and a $20 Solid-State Relay (SSR). That with a bit of wire and some modification of the oven circuit get you a superior soldering system. JLCPCB charges less than $10 for a stainless-steel silk screen and the syringe of solder paste while a bit pricey goes a very long way. The ramp and soak temperatures are not that critical. You can get something that works in a few tries after consulting the manufacturer's suggested temperature profiles. The results will be faster and better than anything you can do with a soldering iron.

I've built 4 or 5 ovens over the years as I used to work on Solid Oxide Fuel Cells as well as metallurgy heat treating projects. With the toaster oven, the wiring is simple and you can find lots of on-line references for building ovens of any kind. The hardest thing is to program the controller as the button sequences can be difficult to get right. The manuals are best described as terse. RTFM carefully and take notes of button sequences. It is lots of fun to watch the flux melt, then see the solder melt and turn silver and even see the chip align up with the pads.

The solder masks are easy to use and the protective support boards they ship in are great working surfaces. Use a couple of spare PCBs to line up two edges of the PCB to be screened. Use masking or paint tape to keep things in place. Line up the metal screen taking care that all points are aligned and tape this down on one edge so it can be flipped up. Add a bit of solder paste from the syringe and spread it over a small area of the mask with a credit card or hotel room key or some such tool. Filling the holes completely is almost too much solder but make sure you don't skimp any areas. Use a very light fill for VSSOP or other fine pitch devices. The mask will add too much solder and bridges between pins will occur. These may not even be visible. Removing them can be difficult and require high heat on copper braid. Clean the boards and masks with alcohol.

LTspice

LTspice is Analog Devices circuit modeling program. It is free and comes with both very simple and complex models. It is best used to model subcircuits and not entire project schematics. I spent a few weeks understanding the equivalent resistance and inductance of capacitors and found LTspice very useful in understanding their impact on the frequency response (phase and amplitude) of actively damped op-amp circuits. Texas Instruments has their own version (TINA?) but is not as popular and on-line user information is scarcer.

The Money, Space, Time Continuum

Since I am an over-educated codger, I can well afford what is a relatively cheap hobby in comparison to say a fancy Bass boat with a nice trailer and a big pick-up to haul it all around. Given the adventure of learning analog electrical engineering from both a theoretical to hands-on experience, I'd say I am enjoying the whole process. The hobby money out flow for boards and components is maybe $200/mo. I have approximately $5000 in test equipment, tools computers and soldering equipment that has been accumulated over about two years. But if I were 20 something this would not be very sustainable in part because one needs a place to do it. It can be very small but it has to be yours and everybody else can fuck off. Do not touch my shit! OK! "Nein gefinger gepoken." Nor would I have the money, but my Christmas and Birthday lists would be highly specific as well as a subtly organized into group gift giving schemes. And at last resort, as dad would say, "GAFJ." So anyhow, I recommend electrical engineering as an excellent hobby. I would prefer mountaineering but that just is no longer in the cards.