Sorcerer VCLFO kit assembly instructions

Note: This kit is relatively simple and straightforward; it makes a good beginner's project or a quick build for experienced DIYers. That said, this build document does assume a basic level of soldering competence—if this is your first project, I highly recommend reading/watching some tutorials and if possible, practicing on some scrap PCBs before beginning.

Step 1: prepare your workspace
Gather your tools and materials. All the supplies you need to build the module are included with the kit. Additionally, you will need a soldering iron and some solder, as well as a way to power your module. A multimeter is not strictly required, but it's so helpful for DIY in general (checking your work and troubleshooting) that you may as well get one anyway. An oscilloscope is not required, but can be used to test the outputs before hooking up to your modular.

Double check the included supplies against the BOM provided on GitHub. If you're missing any parts, please don't hesitate to contact me (sorcerersynth@gmail.com or @jepyangmusic on instagram).

Step 2: solder back-side components
As a general rule, you will want to solder components according to their height off the PCB. This is not a strict requirement, but it makes the process easier.

With this in mind, begin by soldering the resistors. It's easiest to keep track of your work if you place all of one value, double-check, and solder, then move on to the next value. (It's also a good idea to double check the value of at least one resistor in each baggie using your multimeter—I assemble kits with care but everyone makes mistakes.) Resistors are not polarized, which means they can be placed in either direction, but your module will look nicer and be easier to troubleshoot if you make sure to place them in a consistent direction. Depending on how you're holding the PCB (laid on a table vs in a PCB holder), you may find it easier to solder the leads right from the component side, or to bend the leads, flip the board, and solder from the other side. Either works, but make sure your solder joints look nice before moving on. In total, you should solder 15 resistors.

Now solder your diode. This is a polarized component—it *must* be placed in the correct orientation. Match the stripe on the part with the stripe on the PCB footprint. See photo at the end of this document to check your work.

Next solder your capacitors. Start with the 100n caps. These, like the resistors, are not polarized—I like to place them so that the label on the part will be easily legible once the module is built. Then move on to the two 10u electrolytic capacitors. These *are* polarized—the longer leg goes through the pad marked +, and the negative mark on the cap itself is aligned with the white half of the PCB footprint. Check your work against the image at the end of this document.

There is one transistor to solder. You'll need to bed the legs slightly to fit the triangular footprint. Make sure the flat edge of the transistor aligns with the flat edge on the footprint (it should face the edge of the PCB). Again, check your work against the image at the end.

Now is a good time to solder the IC socket. Make sure the u-shaped notch at the end of the socket matches up with notch in the footprint (guess what—you can check this against the image at the…you get it). Electrically it makes no difference, but you're less likely to place the actual IC incorrectly if the socket is facing the right direction. My favorite way to solder IC sockets is to place the socket and bend two pins at opposite corners to keep the socket from falling out. Solder each of the bent pins, but then heat each joint while gently pressing on the socket from the other side—the idea is to get the socket flush against the PCB. Be careful pushing on the socket before the solder has melted—you can inadvertently push the pin up through the socket instead of down through the PCB. Once you have the socket seated nicely against the PCB, go ahead and solder the rest of the pins (and reflow the two bent ones for good measure). Don't place the actual IC yet.

Last thing to place on this side of the board is the power header. This is a keyed header, meant to prevent you from plugging in a power cable backward, so it is very important you place it in the correct orientation. On the side of the header with the notch, you'll see a small arrow in the plastic—this arrow corresponds with pin 1, which has a square pad and is marked in tiny type on the PCB saying "redstripe."

Double check your work against the image at the end of the document. In particular, check the orientation of your diode, transistor, electrolytic caps, IC socket, and power header. The resistors included in your kit may be a different color but as long as you put the correct values in the right spots it wont matter.

Step 3: solder front-side components
It's time to flip your board over and solder the control components.

My favorite way to do this is to place all five components (two pots and three jacks) after removing all the nuts and washers. Don't solder them yet. The pots will kind of "clip" into the PCB while the jacks will be loose and fall out if you flip the board. Fit the panel carefully so that the components come through their respective holes, and then fit the washers to the pots and screw the nuts onto the pots and jacks (careful not to overtighten the nuts on your jacks—solidly finger-tight is all that's needed). The clips on the pots should hold the whole assembly together. Check the front panel—do all the jacks and pots seem nicely aligned? If not, loosen the offenders and wiggle them components into a better position before tightening it up again. Once you're happy that everything is aligned nicely, you can go ahead and solder the pots and jacks without removing the panel.

Step 4: final checks
You now have all components soldered, so give yourself a pat on the back! Before you install into your rack and power up, you'll want to do some quick checks (this is the kind of thing you should do on every module you build).

First off, put your multimeter into continuity check mode. Touch the two probes together—your multimeter should beep to let you know they're connected. Check for continuity between all your power rails; easiest place to do so is at the power header. The two -12V pins at the left side (marked with "redstripe" on the PCB) should have continuity with one another but not with any of the other pins in the header. The six ground pins in the center should have continuity with one another but not with the two -12V pins on the left or the two +12V pins on the right. And the +12V pins on the right should have continuity with one another but not with any of the other pins. If you don't get any beeps where you shouldn't, you can move on with the build. If you get continuity between rails that shouldn't be connected, stop and check your work closely. Make sure there are no solder bridges or splatters anywhere. DO NOT CONTINUE UNTIL YOU FIX THE ERROR.

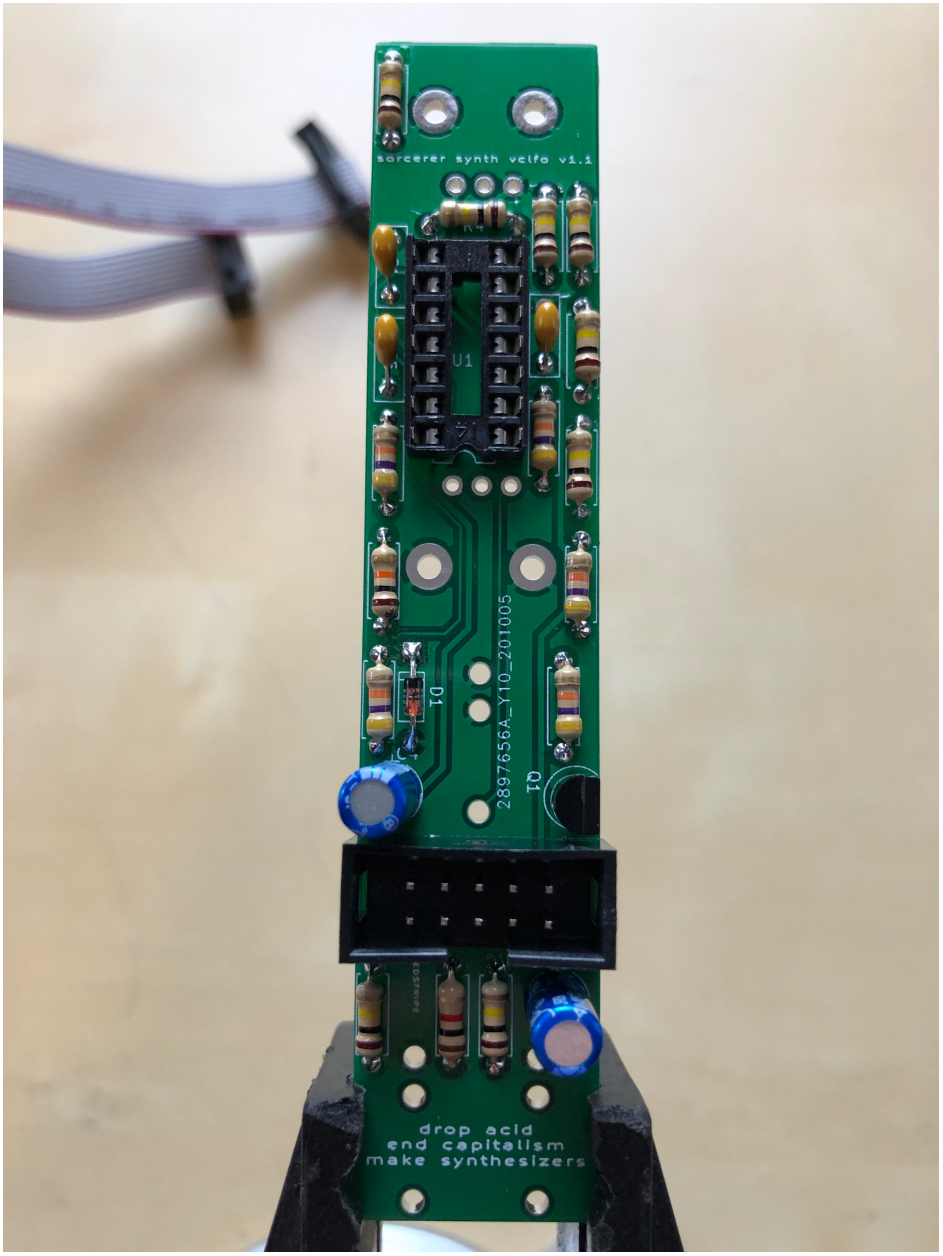
(Checking the power header is usually enough but if you want to be extra sure… Before inserting the TL074 IC into its socket, go ahead and power the module (usually best to do this with a different power supply than is running your $5000 rack, but sometimes this cant be helped). Being very careful not to short any pins with your multimeter probes, check the voltage between ground (the "clip" legs of the pots are a convenient spot) and pins 4 and 11 (the center pins on either side) of the IC socket. You should get +12V and -12V (or something quite close) respectively. This is a simple enough module that this test is not really necessary, but it's a good habit to get into, especially once you move on to more complex modules that use more expensive ICs.)

Now you are ready to go ahead and insert the IC. You will likely find that you need to bend the legs slightly to get a good fit in the socket—simply place the legs against your workbench and bend all seven legs per side at once. Make sure you align the u-shaped notch on the IC with the similar notch on the socket and PCB footprint. Push the IC into the socket firmly to make sure its fully-seated.

As a last precaution, take a close look at the board. Do you see any obvious solder bridges? Splatters? Tiny flakes or balls of solder? Any blobby or under-soldered joints? None of the IC's pins were bent while inserting it? Anything else seem amiss? It's good to fix any possible issues before moving further.

Step 5: finished!
Once you've double checked for shorts and other errors, you *should* be finished! Plug your module in and try it out! The top pot controls the base frequency of the LFO. The middle jack is a CV input and the second knob attenuates whatever CV you send to the jack. The two jacks at the bottom are your outputs. Check these outputs with an oscilloscope, an "LED jack," or simply by plugging them into a modulation output of another module. (Note: this is a pretty simple circuit, so don't be alarmed if the triangle wave isn't exactly linear or perfectly balanced between rise/fall; this is expected behavior.) If you're getting good outputs….that's it! Enjoy your new LFO, make some fun wub-wub sounds!

“The image at the end of this document”



Double-check your work! Pay close attention to the orientation of the diode, the transistor, the two electrolytic caps, the IC socket, and the power header! Your resistors will look different and have different stripes on them but will be the same value (these are 5% resistors I had laying around for my test build, your kit comes with 1% metal film resistors).