Nobody can stop you from printing circuit boards

Most people’s relationship with technology is distant or obscured. The average consumer does not have direct control over their tech — it is usually closed off to them, a little box given to them that occasionally breaks and eventually dies.

But what DIY freaks, keyboard nerds, and synth weirdos have known for a very long time is that it is trivial and, in fact, very fun to just get a stack of PCBs of your own design sent to your house. What’s more, with a little know-how in KiCad, you can actually customize your circuit boards and turn it into a personal canvas for fun art, and there’s not a damn thing that God, the Pope, or the Man can do to stop you. Let’s dive in.

a:hover]:text-gray-63 [&>a:hover]:shadow-underline-black dark:[&>a:hover]:text-gray-bd dark:[&>a:hover]:shadow-underline-gray [&>a]:shadow-underline-gray-63 dark:[&>a]:text-gray-bd dark:[&>a]:shadow-underline-gray”>Photo by Amelia Holowaty Krales / The Verge

a:hover]:text-black [&>a:hover]:shadow-underline-black dark:[&>a:hover]:text-gray-e9 dark:[&>a:hover]:shadow-underline-gray-63 [&>a]:shadow-underline-gray-13 dark:[&>a]:shadow-underline-gray-63″>*Tetsuo, the Keyboard man.*

a:hover]:shadow-highlight-franklin

dark:[&>a:hover]:shadow-highlight-franklin [&>a]:shadow-underline-black dark:[&>a]:shadow-underline-white”>Alright, you’ve sold me. How do I do this?

In order to get PCBs made, you are going to want to select a project, usually a GitHub repo for the circuit board of your choice. Then you are going to have to locate the Gerber files. Set up your account and clone the repo, or if you aren’t comfortable using GitHub (understandable), you can use tools like DownGit to just download it manually.

a:hover]:text-gray-63 [&>a:hover]:shadow-underline-black dark:[&>a:hover]:text-gray-bd dark:[&>a:hover]:shadow-underline-gray [&>a]:shadow-underline-gray-63 dark:[&>a]:text-gray-bd dark:[&>a]:shadow-underline-gray”>Photo by Amelia Holowaty Krales / The Verge

Because I am a dweeb, here are a handful of keyboard (predominantly ergonomic split keyboards) to start with that have PCBs you can just download, print, and doodle on. These ones don’t require too many tiny surface-mounted parts and are easy to solder in your own home, although many companies like JLCPCB will sell you parts and do assembly if you have the files, but that’s outside the scope of this post.

Here’s some classic boards:

- Corne by Foostan
- Sofle by Josef Adamcik
- Nijuni by krikun98
- Sweep (based on the Ferris) by David Barr
- Cantor by Diego Palacios
- LMN-3 (a *DAW* in a box) by Fundamental Frequency

You are going to want to download KiCad. It’s a free, open-source electronics design suite, and a huge percentage of people that make PCBs online use it. Take the project from the repo of your choice. Draw or select your image. You need something that pops visually in black and white, so a high-contrast image or something with really nice patterns. Usually, I try to dither and resize my images in something like Photoshop before bringing them into my project, that way the silkscreen isn’t too complicated and doesn’t need to be resized in KiCad.

a:hover]:text-gray-63 [&>a:hover]:shadow-underline-black dark:[&>a:hover]:text-gray-bd dark:[&>a:hover]:shadow-underline-gray [&>a]:shadow-underline-gray-63 dark:[&>a]:text-gray-bd dark:[&>a]:shadow-underline-gray”>Photo by Chris Person / The Verge

Now that you have your image, open KiCad and select the “Image Converter” on the right-hand side. From there, you are going to want to select “load bitmap” and load whatever image you wanna print on your PCB. You are going to need to take into account the maximum DPI that your PCB manufacturer can handle (300 dpi is usually a good rule of thumb). For me, this process is always a lot of trial and error, but don’t get frustrated if the program crashes.

You can make liberal use of the various tabs to see how the image is going to look in black and white and invert the image if it does not work with your silkscreen. Most of the time, you are going to be working with a white silkscreen, unless you are working with white FR4 in which case the silk will be black, so keep that in mind when establishing the negative space on your art. You can even up the contrast in KiCad to make the file a little more legible. When it looks good, you can save it to KiCad’s footprint library. You can always come back here if the silkscreen isn’t perfect when you drop it on

your board, and you probably will.

Load up your project. In this case, I am going to go with the Cantor, a wonderful PCB to start with for a number of reasons: it uses very cheap STM32 “blackpill” controllers that cost next to nothing on AliExpress, it doesn’t use hot-swap sockets so you can see your art clearly, and it doesn’t require a case which reduces the number of parts you need to order or make.

a:hover]:text-gray-63 [&>a:hover]:shadow-underline-black dark:[&>a:hover]:text-gray-bd dark:[&>a:hover]:shadow-underline-gray [&>a]:shadow-underline-gray-63 dark:[&>a]:text-gray-bd dark:[&>a]:shadow-underline-gray”>Photo by Chris Person / The Verge

Once your PCB is loaded up, you are going to press “O” or click the “load footprint” button on the right-hand side of the toolbar. This brings up all the “footprints” in your library, which includes any silkscreens you may have previously generated. You can use the filter up top to quickly narrow down your choices to the art you just generated. If your silkscreen is too big or small for the area you have selected, refine it in one of the previous steps and try again. If you see something that looks like “G***,” double click your silkscreen and select properties, then untick “visible” to make that part of the footprint invisible. The layers you are going to be editing on are usually called “F.Silkscreen” and “B.Silkscreen” for front and back, respectively. If you want the silkscreen on the other layer, press “F,” and it will flip the silk to the other side of the board. This is handy if you want an image on both sides of a board. Place your silk on a relatively clean and empty part of the board or get creative — it’s design! To preview how the board is going to look, you can open up the 3D viewer in KiCad and rotate your board around.

a:hover]:text-gray-63 [&>a:hover]:shadow-underline-black dark:[&>a:hover]:text-gray-bd dark:[&>a:hover]:shadow-underline-gray [&>a]:shadow-underline-gray-63 dark:[&>a]:text-gray-bd dark:[&>a]:shadow-underline-gray”>Photo by Chris Person / The Verge

Now that you have your art on a board, you’re going to have to find a company that manufactures PCBs. I’ve had luck with PCBWay and JLCPCB, but there are tons of them out there like OSH Park as well as online tools you can use to compare prices.

When you are done, you are going to want to generate your gerber files using KiCad’s fabrication options. If you are ordering from JLCPCB specifically, there are actually handy plugins for JLCPCB and PCBway that you can use to simplify the entire process. When you are done, just zip the files up and upload the compressed archive to your manufacturer of choice. Depending on who you buy from, it should generate a preview where it’ll show you what the board looks like and give you several options for materials, number of layers, colors, and more. I love a board that pops, so I tend to go with Blue FR4 with a white silkscreen or White FR4 with a black silkscreen, but the options vary from manufacturer to manufacturer. OSH Park, for example, offers a beautiful black “After Dark” PCB with a clear solder mask (the polymer applied to the top of copper traces – this means you’ll be able to see the copper under it). It’s definitely a premium, but it’s very handsome, and you can do some very neat stuff with it.

a:hover]:text-gray-63 [&>a:hover]:shadow-underline-black dark:[&>a:hover]:text-gray-bd dark:[&>a:hover]:shadow-underline-gray [&>a]:shadow-underline-gray-63 dark:[&>a]:text-

gray-bd dark:[&>a]:shadow-underline-gray”>Photo by Chris Person / The Verge
Okay, so in my example, I am getting 20 PCBs in purple FR4 with lead-free HASL from JLCPCB, which is enough to make about 10 boards before you factor in parts. As with most things, the more you buy the greater the discount, so consider making a bunch of these for you and your friends. The rough cost here is about 25 bucks plus shipping. If you really need it now, you can pay a premium for something like DHL. I personally tend to go with the mega-cheap shipping because I am just doing this for fun and don’t care if it takes a few months to get here. Your mileage, of course, may vary!

When it arrives, it’ll come vacuum-packed in a little box and be ready to solder and assemble. If you got a keyboard, you are going to need keyboard parts, so it makes sense to order those on AliExpress at the same time so they get there whenever. Alternatively, you can support one of the many fine keyboard suppliers. But of course, this goes beyond simply making cheap keyboards. I have gotten domesday duplicators made along with multiple arcade sticks and trackball PCBs (for complicated boards, you may need some soldering and assembly done at the factory). The examples I have laid out are fairly simple. I just like putting Pop Team Epic characters on my gadgets, but if you master tools like KiCad, you can not only print silkscreens, you can create elaborate art that integrates the existing copper layers into the art itself. (The Sweep, Urchin, Smitheii and the Draculad are good examples of this). And if you’re actually *good* at drawing and design? You can become a force to be reckoned with.