Milling Custom Circuit Boards on the Roland Monofab SRM-20 with KiCAD and MODS

This document is designed to give a short summary showing how to create custom circuit boards using tools widely available at Wheaton College and other makerspaces. It is not intended to be a full training manual.

Basic info on what a PCB (printed circuit board) is, and how it works:
https://learn.sparkfun.com/tutorials/pcb-basics
https://www.autodesk.com/products/fusion-360/blog/an-introduction-for-electronics-beginners-printed-circuit-boards-from-10000-feet/

This tutorial assumes you start with a circuit board designed in KiCAD. It does not show you how to do that, but here are some great tutorials on the subject:

http://fab.academany.org/2019/labs/barcelona/local/wa/week6extra/kicad/

https://learn.sparkfun.com/tutorials/beginners-guide-to-kicad/

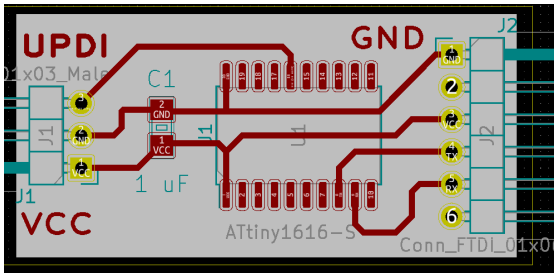
This tutorial covers single-layer boards only.

This is only one of many ways to create a circuit board. For a diagram showing many possible workflows and their strengths and weaknesses, see

https://docs.google.com/drawings/d/1_y4-QAegV5s56sJBFXjrf8ToHd4KgXSRxVFkgnWxnKY/edit

Creating an SVG file from KiCAD

We'll be starting with a completed `.kicad_pcb` file from a KiCAD project. Here are some tips to make your life easier:



1. Whenever possible, use fairly **wide traces** (0.5 mm / 10 mils) and at least 0.4 mm **clearance** between traces. You can change the defaults in Board Setup / Design Rules / Net Classes, or change the trace width afterward in Edit Track and Via Properties.
2. Use the Fab Academy KiCAD **parts library** (https://gitlab.fabcloud.org/pub/libraries/kicad). Follow the installation instructions there carefully! Whenever possible, use fab library parts: we have many of these on hand in the lab.

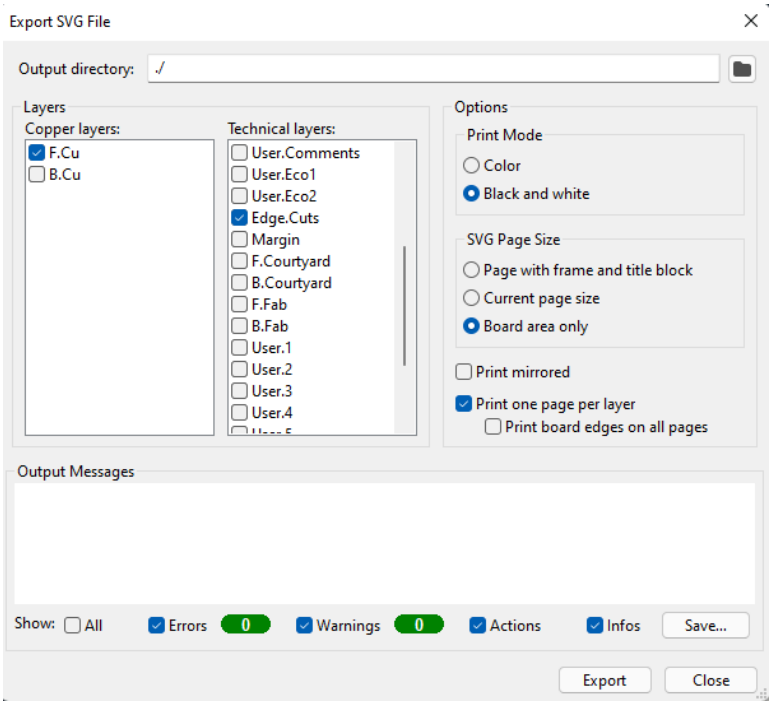
3. Draw your board outline shape as a polygon or rectangle on the **Edge.Cuts** layer (gray rectangle in the board image above). Then right-click it, choose “Properties”, and make it a “Filled Shape”.
4. Draw a box on the “**Margin**” layer that’s slightly larger than the size of your board outline shape on Dwgs.user. The small gap is needed so MODS can properly detect the board outline.

Choose File / Plot … For MODS, you will want to export two SVG files: the circuit traces file, and the board outline file.

Set Plot Format to “SVG”

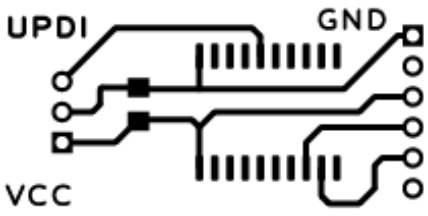
Export the front front copper layer (F.Cu) and the Edge.Cuts layer at the same time. Select all of the following **(critical)**!

- “Black and white”,
- “Board area only”,
- Uncheck “Print Mirrored”
- Check “Print one page per layer”
- Uncheck “Print board edges”



F.Cu contains your traces: rename the F.Cu file `traces.svg` or something similar. The Edge.Cuts layer contains your board outline: rename this file `outline.svg` or similar.

If your board has through-holes that must be cut, there are some extra steps. To generate your board outline file, export the three layers again, this time in a single file. Second, edit this file in a vector drawing program to move the board outline shape to the back. (“Object / Lower to Bottom” in Inkscape, “Object / Arrange / Send to Back” in Illustrator.) This will make the holes visible, so the Roland will cut them. Save this as your outline.svg file.



`traces.svg`

`outline.svg`

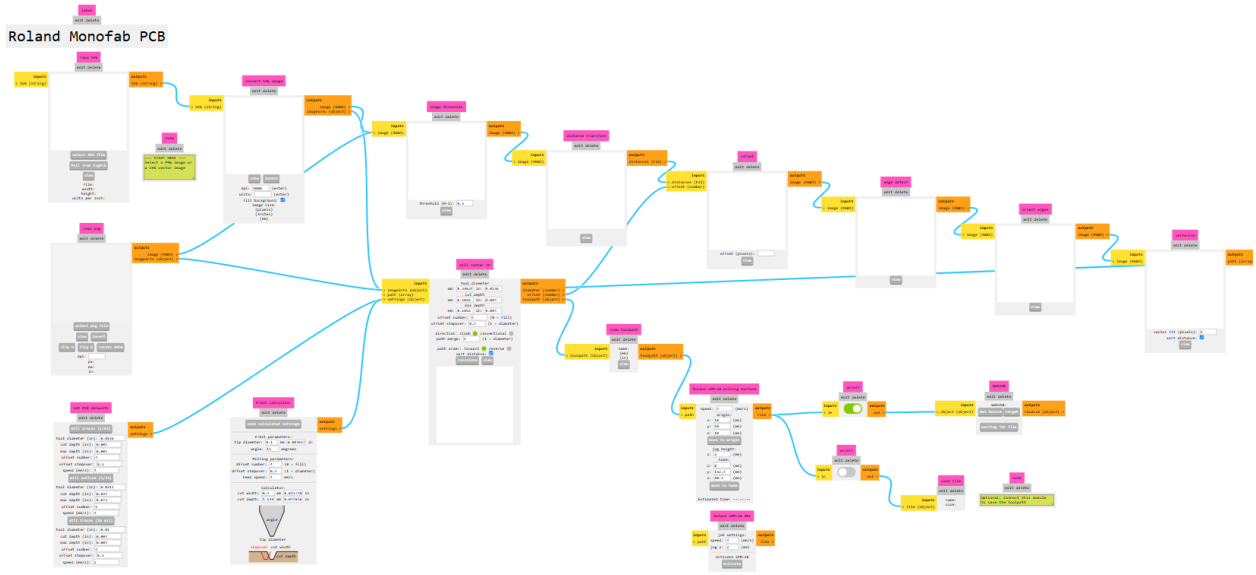
Creating Roland Mill Files with MODS

MODS is a multipurpose fabrication file conversion tool associated with Fab Academy. We will use it to convert our .SVG files to .RML files, which will be sent to the Roland Monofab mill.

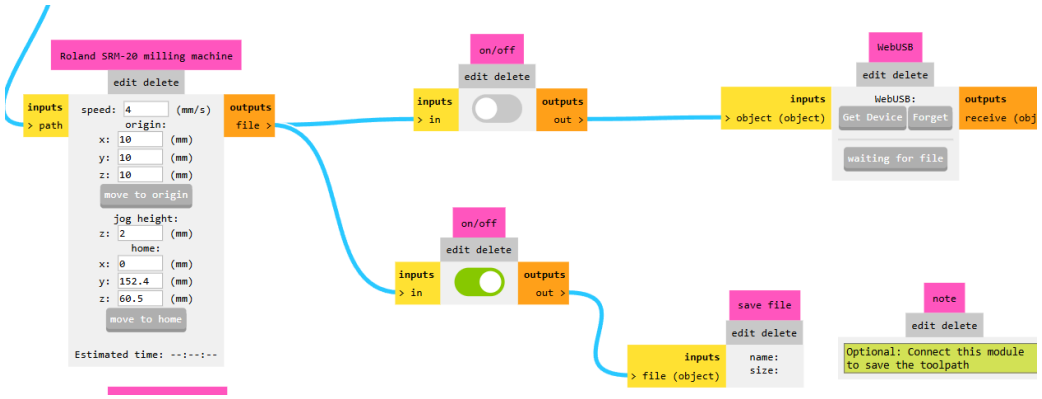
Go to https://modsproject.org/ .

Right-click anywhere, and choose “programs”, “open program”, and then **“Roland / mill / SRM-20 mill / mill 2d PCB”.**

This bewildering rat’s nest is a program to trace out the edges of our SVG file and create a milling toolpath. You could use *any* SVG file, not just one created in KiCAD, with MODS, even one hand-drawn in Illustrator or Inkscape.



To make the program save the toolpath as a file, rather than trying to send it directly to a Roland mill connected to your computer, in the bottom right of the program, turn the “WebUSB” switch OFF and the “save file” switch ON.



Next, load up the board traces file. Click on “Select SVG File” in the top left, and select the `traces.svg` file you created in the previous section.

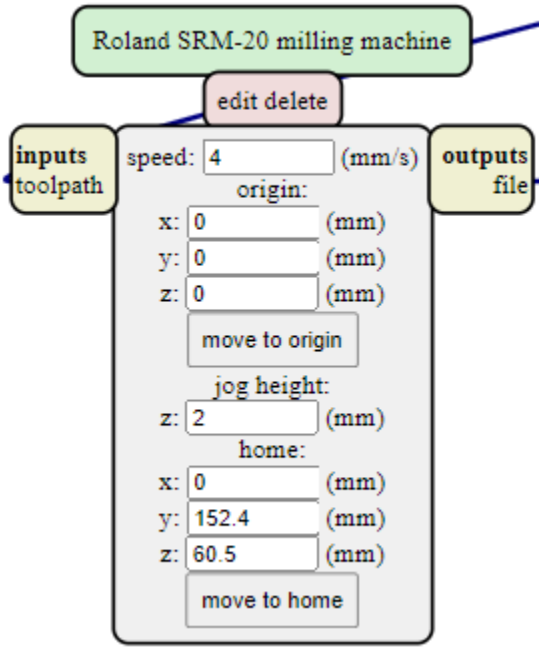
MODS Settings for cutting traces:

Click “mill traces 1/64”. Check that “mill raster 2D” has the following settings:

- Tool diameter 0.39624 mm
- Cut depth 0.1 mm
- Offset number: 4
- Offset stepover: 0.5

Set the following settings in “Roland SRM-20 milling machine”:

- Speed: 4 mm/s
- Origin: **x=0, y=0, z=0**[1]
- Home: don’t change it, but check that **z > 0**!



Now click “Calculate Traces” in “mill raster 2d”. Your computer should download a file named “SVG image.rml” or similar. Rename it “traces.rml” or similar.

MODS Settings for cutting board outline:

Click “mill outline 1/32”. Check that “mill raster 2D” has the following settings:

- Tool diameter 0.79248 mm
- Cut depth 0.6096 mm
- Max depth: 1.829 mm
- Offset number: 1

Set the following settings in “Roland SRM-20 milling machine”:

- Speed: **1.0 mm/s** **(This is conservative, but increase at your own risk.)**
- Origin: **x=0, y=0, z=0**

[1] **Red text** indicates settings that can destroy bits or damage the machine if you get them wrong.

Home: don't change it, but check that **z > 0**!

Now click "Calculate Traces" in "mill raster 2d". Your computer should download a file named "SVG image.rml" or similar. Rename it "`outline.rml`" or similar.

Saving as a Local Program

You can save this modified MODS program to your local computer, so you don't have to re-do the steps above every single time. Right-click anywhere and choose "Programs / Save Local Program". Drag the window to show the top left part of the MODS window: there will be a place there to type in a filename (it's sometimes offscreen.) From now on you can do "Programs / Open Local Program" to read in this custom program.

Handy Tiny Dimension Conversion Table

Fractional inch	Decimal inch	mils	mm	Typical uses
1/64	.016	16	0.40	Tiny endmill
1/32	.031	31	0.79	Less tiny endmill
1/16	.062	62	1.57	Board cutouts, common board thickness
1/20	.05	50	1.27	Common surface-mount pin spacing
1/10	.1	100	2.54	Common through-hole pin spacing

Roland Monofab Operation

Time to fire up the mill and get cutting! But first, an important safety concept:

Air-Cut First for Safety!

You never know what's going to happen the first time you run a CNC job. A bad CNC job will tell the mill to eat itself, and many machines are happy to do so. It's always safest to do an "air cut" first, where you set the Z-axis deliberately too high, so the machine cuts through the air above your board, rather than cutting anything real. Once you know it's working, lower the bit and do a real cut. This tutorial doesn't walk you through the "air cut" process.

Anyway, let's get started on a real cut. The **power button** is on the top back right corner. To operate it, you need to start **VPanel** on the connected PC.

Step 0: VPanel Setup

Click the "**Setup**" button in VPanel. For Command Set, choose **RML-1 Code**[2]**.** For Units, choose "**mm**".

Step 1: Material Prep

Use double-stick tape to stick your PC board material to the wooden spoilboard in the Roland. **Make sure there's double-stick tape under the area where the board will be cut out, if you created a custom board cutout (Option 2 of Step 5 above.) If not, the board will break loose and fly away.**

Step 2: Tool Prep

Use the allen wrench to loosen the set screw on the Roland's collet (the part that holds the cutting tool.) **Insert your new tool fully, and tighten the set screw**. Now, use the arrow keys in VPanel to **move the bit close** to (but not quite touching) the board material, **loosen the set screw** so the bit falls onto the board, and **re-tighten**. Now we know the bit is just touching the material. (But don't forget to air-cut first!)

Step 3: Set the Origin Point

Hit the "Z" button in the top right corner of VPanel to **zero the Z work coordinate** at the top of the board material. Now raise the bit in Z, for safety.

---

[2] Always check this setting! Users who use FlatCAM to run the Roland mill will switch it to "NC Code".

Move the bit to the position you want to use as X=0, Y=0. This corresponds to the bottom left corner of your board outline in MODS. Hit the “XY” button in the top right corner of VPanel to **zero the Y and Z work coordinates**.

Step 4: Run the Job!

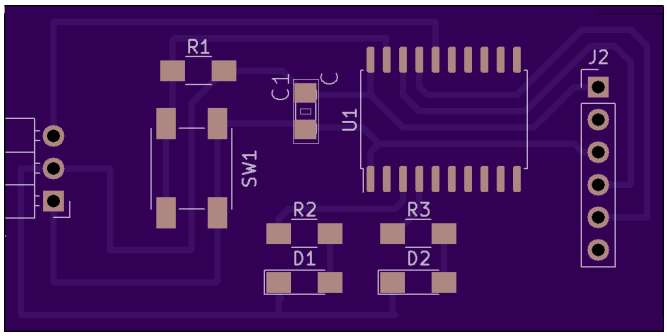
Hit the **Cut** button, and choose the .rml files you want to cut. You can do more than one at a time, so long as they use the same tool size.

Step 5: While Cutting

While the machine is in operation, you can **open the front cover** to make it stop instantly, or hit the **pause** button in VPanel to pause temporarily. In VPanel you can adjust the travel speed (**cutting speed**), or the rotation speed **(spindle speed).** In general, if it’s vibrating and making scary noises, slow down the cutting speed. If the edges are charring while working with large diameter tools, slow down the spindle speed -- otherwise faster is better.

Outside Manufacturing

The Roland CNC can create boards that are functional, but they will be difficult to solder to, they can’t be very complex, and they won’t be pretty. A somewhat less DIY approach is to send your manufacturing files to an online circuit board fabricator. Many of these are big-time factories, but there are some services that will make just a few small boards for makers and hobbyists. Usually they group your order with other peoples’ stuff, so the process takes a bit longer but there’s no minimum order size.



The price is cheap ($5 per square inch for three copies), and it’s pretty fast (delivery in under two weeks). You can pay more for faster service. In exchange for your time and money, you can easily make double-sided boards with vias (pass-throughs), that make wiring complex circuits easy. You get a solder mask, which looks nicer and keeps solder from going where it’s not supposed to go. You get gold-plated contacts, which won’t corrode over time. And you get a silkscreen, so you can put custom lettering or art on the board.

I really like **https://oshpark.com/**, partly because their solder masks are purple by default. To use their service with KiCAD, see https://docs.oshpark.com/design-tools/kicad/generating-kicad-gerbers/