# How to KiCAD

John Lawless

3/30/2024

## 1 Introduction

This document will serve as a step by step guide that will allow students to create their own streamdeck using KiCAD. They will learn how to wire, trace, and assemble their own boards.

For the first workshop they will be covering the wiring, they can download all files here. In terms of the tracing, while they should hopefully use their file from the previous workshop, in the event that we have someone that is interested in learning how to trace a PCB, they can download an almost completed schematic. Only 2 wires are needed to be connected in the schematic, however this is on purpose as depending on the orientation of the wires it can cause overlap and may not allow for certain tracing. That file can be found here.

## 2 Installation

### 2.1 Downloading

You can download the latest version of KiCAD here!

### 2.2 Installing KeySwitch Library

For this part of the document it WILL NEED TO BE DONE FOR BOTH WORKSHOPS as it is required to have the footprints and 3D models. Once they click install, they will need to apply pending changes for the library installation to take effect. See the Figure 1 below to see what library to download.
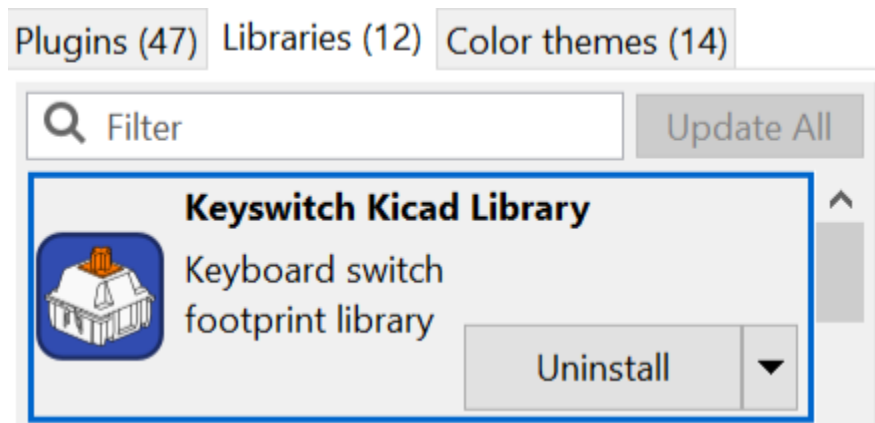


Figure 1: This can be found in the Libraries within the Plugin and Content Manager.

# 3   Wiring

## 3.1   Getting Started With Wiring

Once you download and extract the zip file from the link, you can open the KeyBoardFinal "Project 8.0" File to automatically open the project. Any prompts you encounter on your first installation of the program can just be set to the recommended/default settings.

Double click on "KeyBoardFinal.kicad_sch" to open the schematic editor to begin the wiring process.
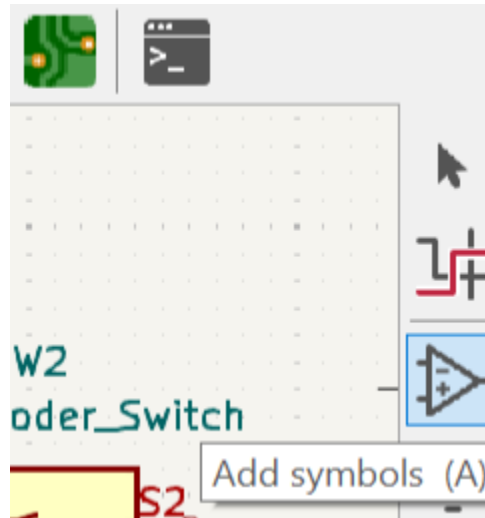


Figure 2: This button will allow you to place parts onto your schematic.

Below is a list of composing of the Name and Number of Components, Component name in KiCAD and the foot print name that we will be using.

- 1x Raspberry Pi Pico

- Pico

- RPi_Pico:RPi_Pico_SMD_TH


- 1x Rotary Encoder

- RotaryEncoder_Switch

- Rotary_Encoder:RotaryEncoder_Alps_EC11E-Switch_Vertical_H20mm


- 4x Switches

- SW_Push

- SW_Cherry_MX_Plate_1.00u


- 2x Resistors

- R

- R_Axial_DIN0204_L3.6mm_D1.6mm_P5.08mm_Horizontal

## 3.2   Editing Symbol with Symbol Editor

For some reason the rotary encoder we are using has the S1 and S2 pins swapped in the footprint. The footprint is used to define the copper connections between physical components and the routed traces on a circuit board. To change this just right click on the rotary encoder and click "Edit with Symbol Editor".
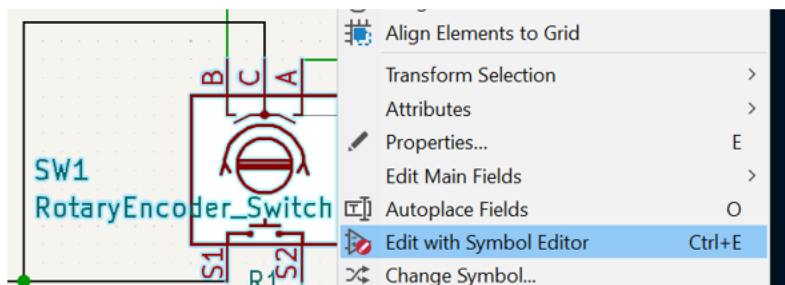


Figure 3: Right click and click "Edit with Symbol Editor".

Next all we have to do is swap the pin name and pin number so that there are no issues when it comes to the tracing part of the PCB design process.
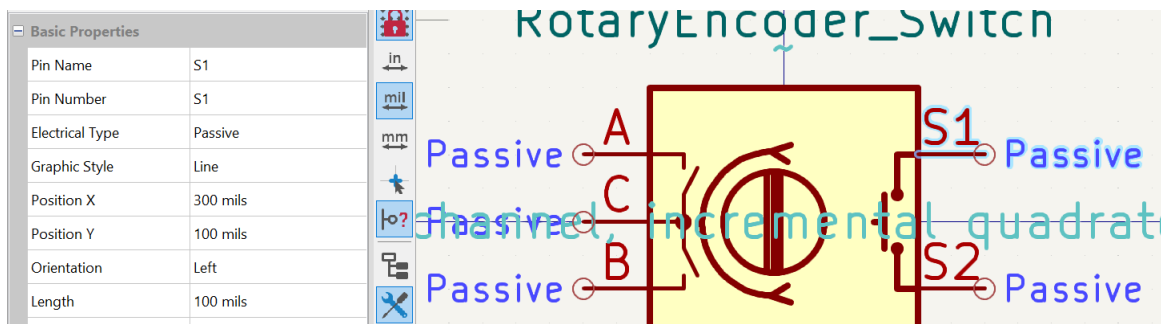


Figure 4:   From this Figure above
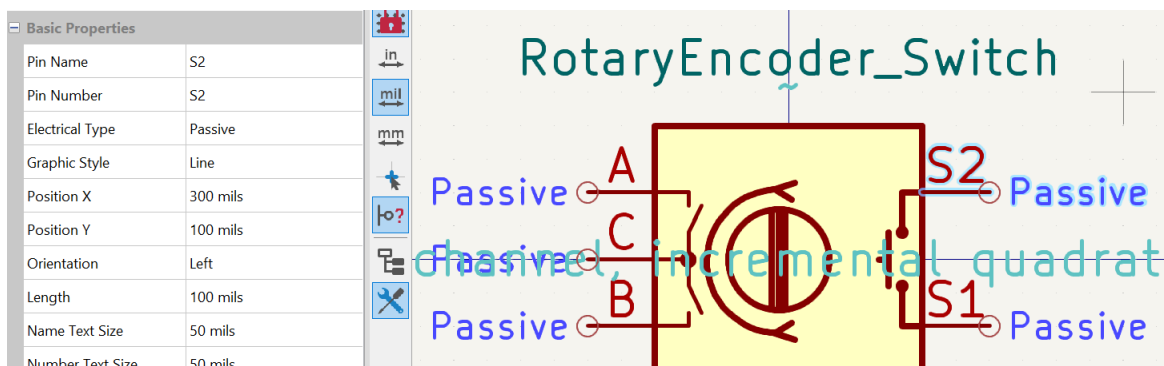To the figure below



Figure 5:   You'll know if you did this correct if the S2 is in line with the A line!

## 3.3 Final Wiring

Below is the final wiring of what the end result should be, follow the connections as in the image below, all you have to click on each connection and match it to its GPIO pin.
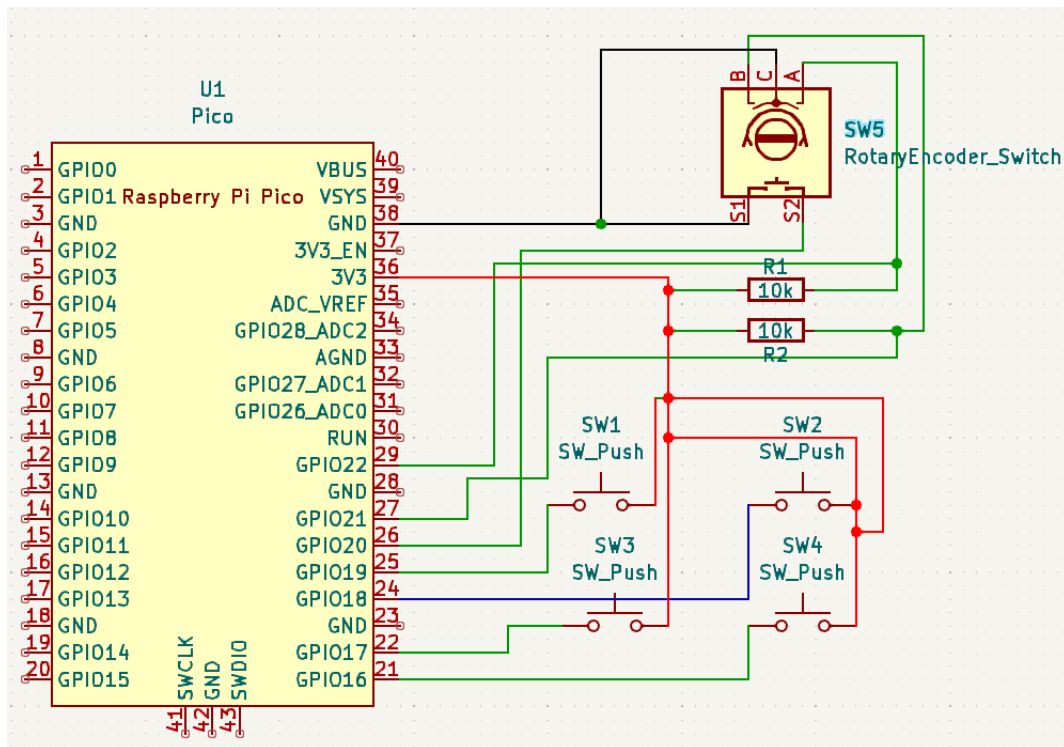


Figure 6: This is how your schematic and wiring should look!

## 3.4 Assigning Footprints

When it comes to assigning footprints, all we have to do is double click a part, click within the footprint field, and click on the small set of books that is at the end of the bar.



Figure 7: This is the button you press after you double click a part.

From here all you have to do is reference the given footprint names to the correlating part.

# 4  How to use the PCB Editor

After we have our finalized schematic, we can then go into the PCB editor by double clicking the kicad_pcb file in the project files. When you open the PCB editor you will have a blank background.
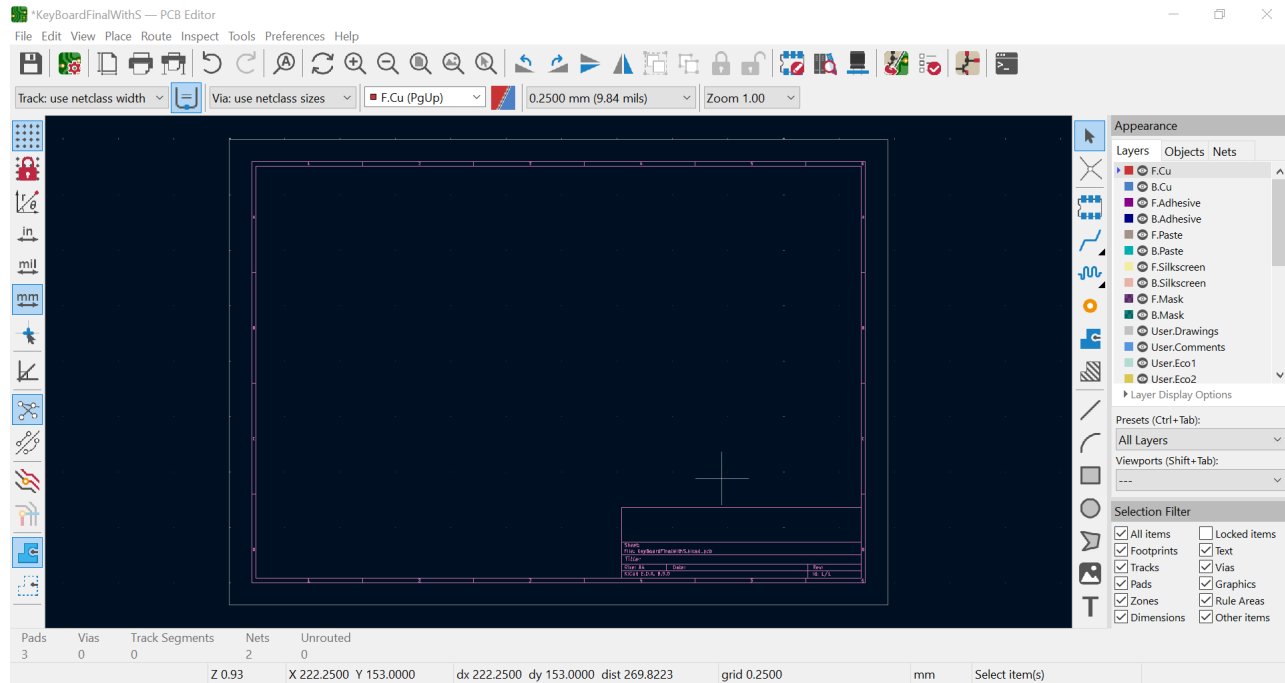


Figure 8: This is how your PCB editor should look when you first open it.

To add our schematic to our PCB editor we can either press the button on the top right that is the right of the check mark and to the left of the capacitor or we can press F8.
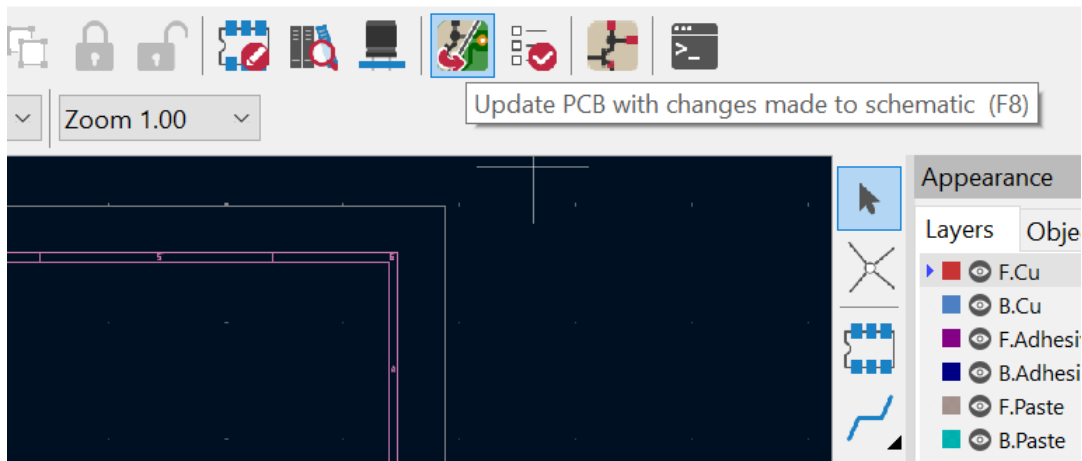


Figure 9: This is where the button is to transfer your schematic.

You'll be prompted with a "Update PCB from Schematic" window, and all you need to do is click on "Update PCB" at the bottom, once you do that, you'll then click on close, and all your components should be attached to your mouse.

## 4.1 Arrangement

You should position all your components as in the figure below to ensure proper tracing can be laid out.
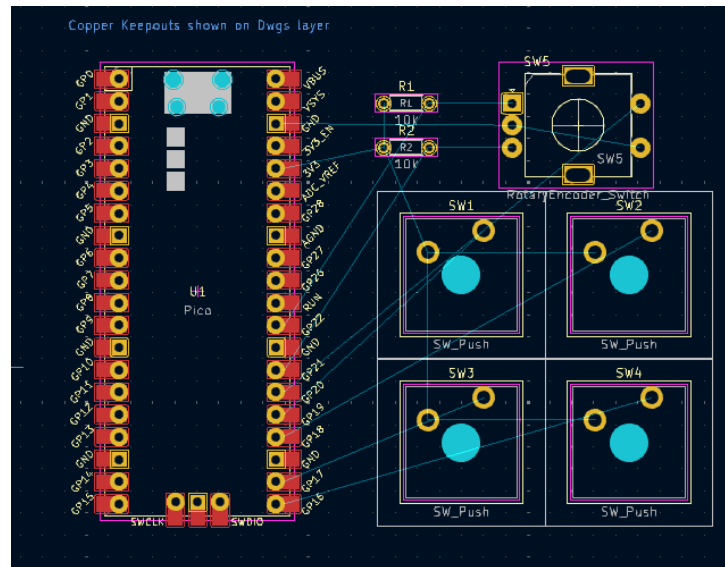


Figure 10: Here is how your PCB Editor should look.

## 4.2 Edge Cuts

Edge Cuts allows us to create the shape of our PCB. If we look on the right side of our PCB editor within the appearance and layers section, if we scroll down we will see a light grey square that says "Edge.Cuts". Select that layer and click on the draw rectangle button and draw a rectangle around the PCB.
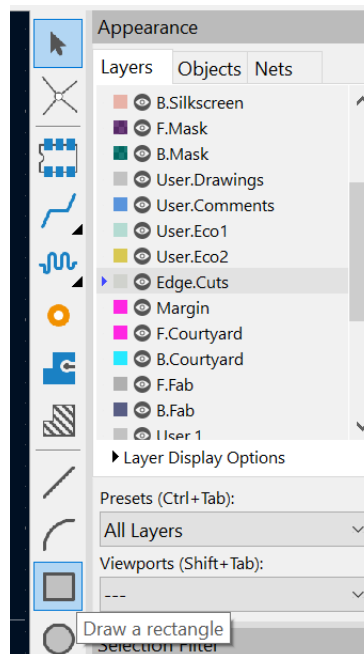


Figure 11: Here is where the Edge.Cuts is located in the Layers tab under Appearance.

## 4.3   Tracing

For tracing ensure that we are on the Front Copper layer or "F.Cu" so we can begin tracing. Next, hover your mouse over any of the yellow pads than have a lines connecting them and press "X".
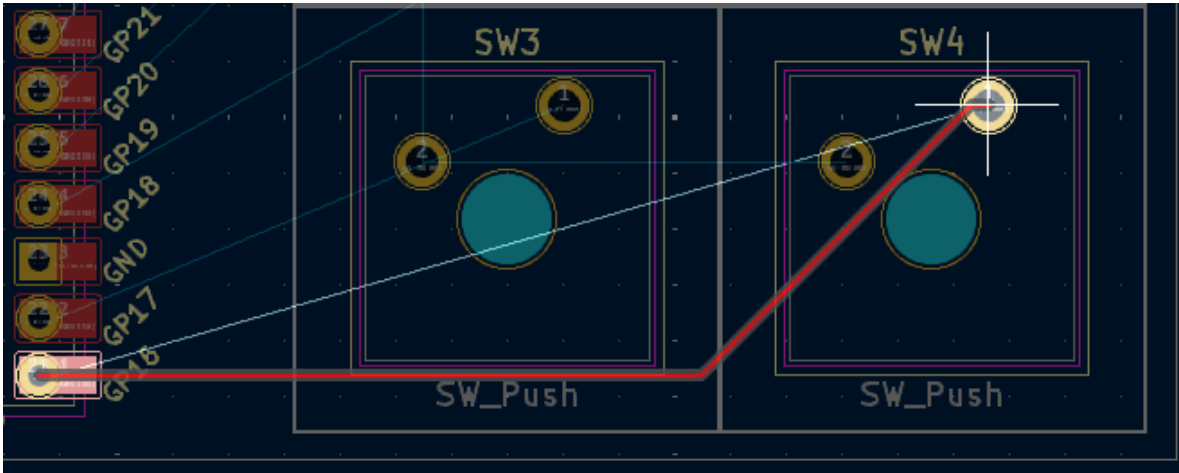


Figure 12: Here we can see GPIO18 being connected to Pin 1 on Switch 4.

Note if there does not seem to be enough room to route your traces, you can switch to the back of PCB by changing your layer to "B.Cu" or by pressing the minus key to switch between the 2. Your routing can may differ from others, but as long as the connections are touching you will be good.
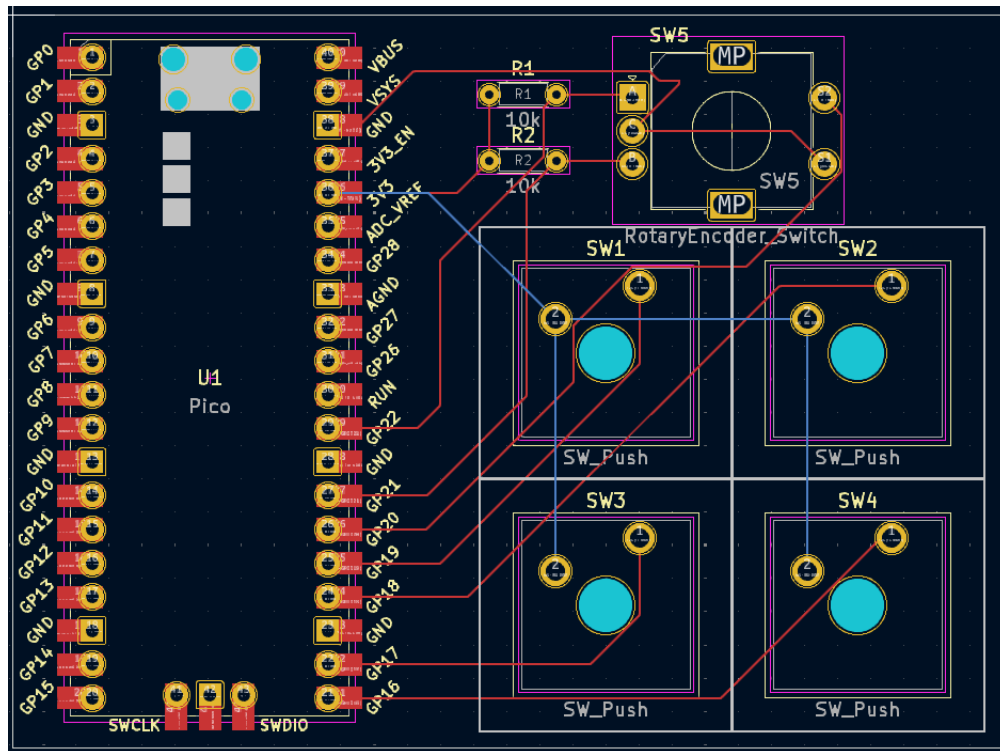


Figure 13: Here is an example of what your PCB could look like.

## 4.4   Ground Plane

Adding a ground plane reduce electrical noise and interference through ground loops and to prevent crosstalk between adjacent circuit traces. To do this in KiCAD, we will look to the right side of our screen and press the "Add a filled zone" button, under the yellow circle.
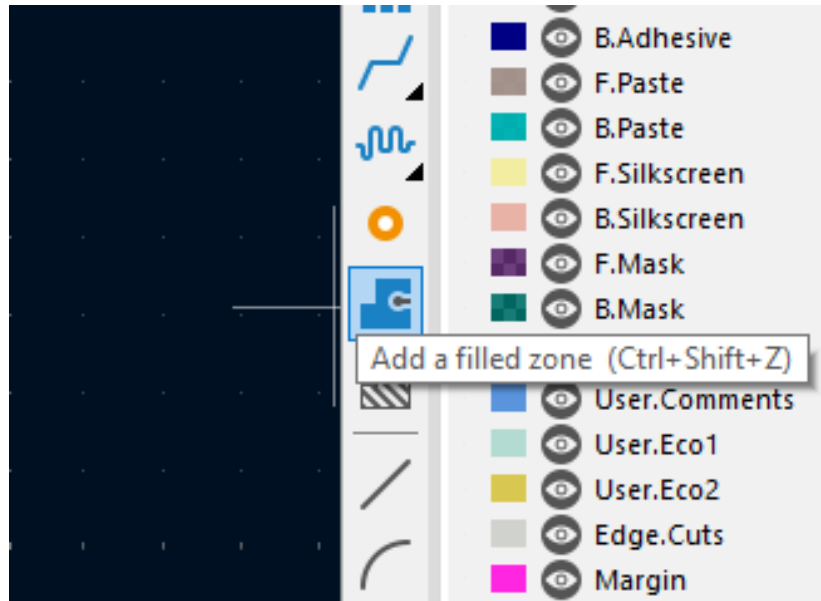


Figure 14: Here is what the button looks like.

Once selected you will want to click the top left of your edge cut/the outline of your PCB. If you used the back of the PCB to do any routing, you'll need to make sure that you select the "B.Cu" as well within the popup. After that click okay and click on each of the remaining corners of the PCB. Once you click the corner you started with you could now see an outline appear around your PCB.
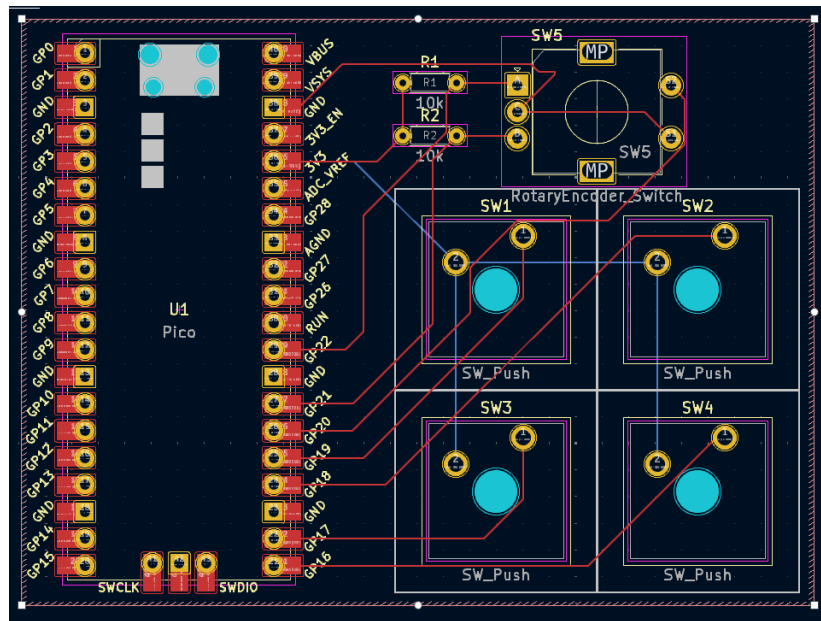


Figure 15: Here is a PCB an unfilled zone.

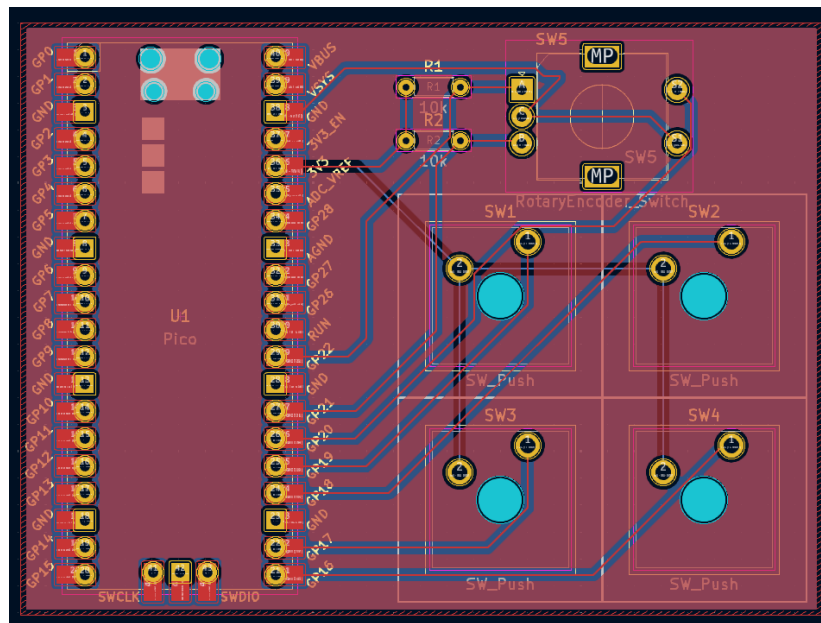All that is left to do it press "B" and you should see your PCB automatically fill in.

Figure 16: This is the end result of adding a ground plane.

## 4.5   3D Models

A nice feature that KiCAD offers is the ability to see a 3D rendering of your PCB. However, we first have to ensure that we are using the correct 3D model with the corresponding part.

   Let's start with the Raspberry Pi Pico. Double click on it and navigate to the 3D model tab. We will then have to locate the 3D model.
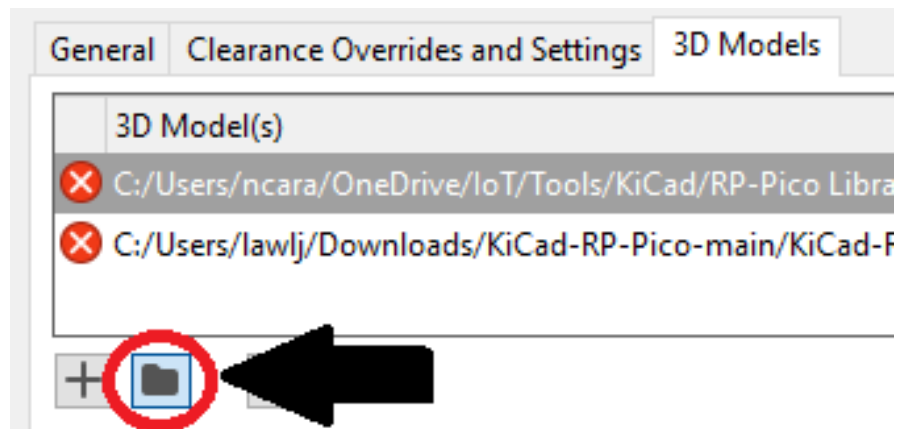


Figure 17: Click on this to enter the file location.

Luckily, KiCAD gives you the option to switch to different available paths, you will need to locate the folder that you are working within as it contains the 3D model for the Pico and the Rotary Encoder.
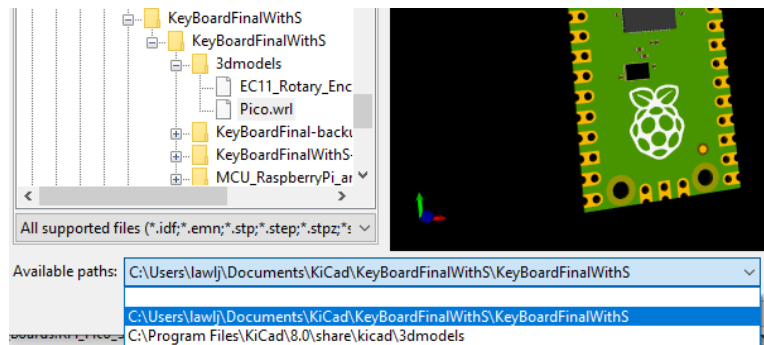


Figure 18: Here is what your file location should look like.

For the keyboard switches however, you will need to navigate to the "3rdparty" folder to find the 3dmodel for the key switches.
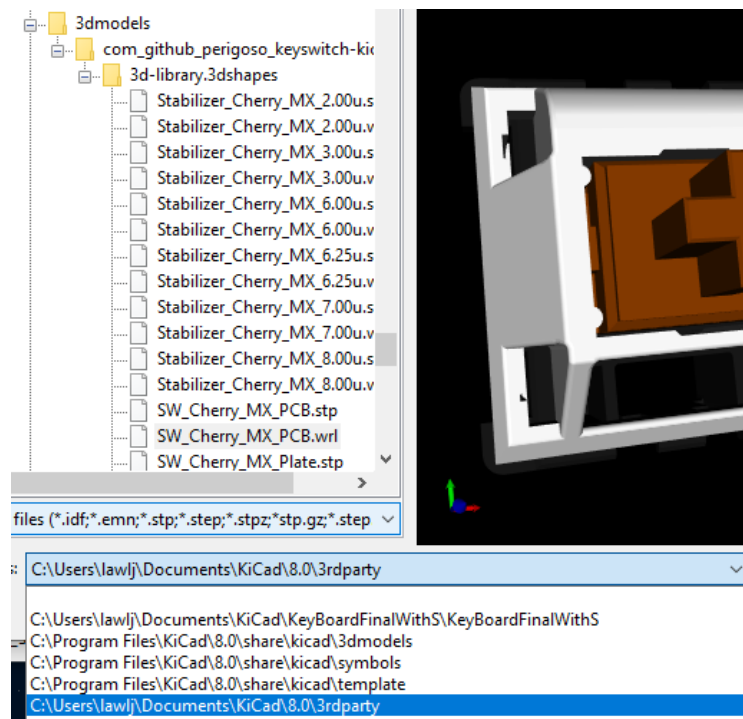


Figure 19: File path and part name

One thing to note about the switches is that it will need an x offset of -2.5 and a y offset of -5.0. Same goes for the rotary encoder as it will need a 90° rotation on the z axis and an x offset of 7.5 and a y offset of -2.5

Once all the models have been assigned properly, we can then press "Alt+3" to see a 3D model of the PCB we made!
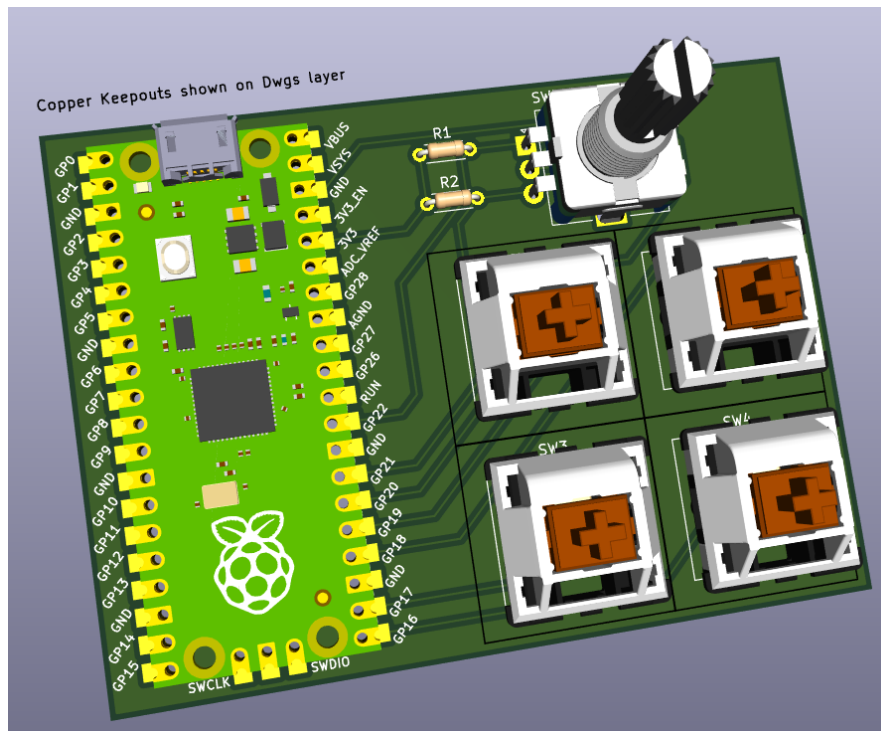


Figure 20: Here you can see a 3D model of your PCB

That concludes the PCB design process from schematic to 3D Model, if you have any other questions please let me know!