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.

2 Workshop Overview

2.1 Workshop 1: Wiring

Students will focus on wiring the PCB. All necessary files can be downloaded here.

2.2 Workshop 1: Tracing

While students should ideally use their files from the previous workshop, this session will also accommodate those interested in learning how to trace a PCB from scratch. An almost completed schematic is provided, which only requires the connection of two wires. This is intentional, as the orientation of these wires can affect tracing and may cause overlaps. The schematic file can be downloaded here.

Key Points:

- Students will learn how to wire, trace, and assemble their own PCBs.

- The first workshop will cover wiring, with all necessary files available for download.

- The second workshop will focus on tracing, with an almost completed schematic provided for practice.

3 Installation

3.1 Downloading

You can download the latest version of KiCAD here!

3.2 Installing KeySwitch Library

For this part of the document, the instructions apply to both workshops, as it is essential to have the footprints and 3D models. After clicking install, they must apply pending changes for the library installation to take effect. Refer to Figure 1 below to see which library to download.

4 Wiring

4.1 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.
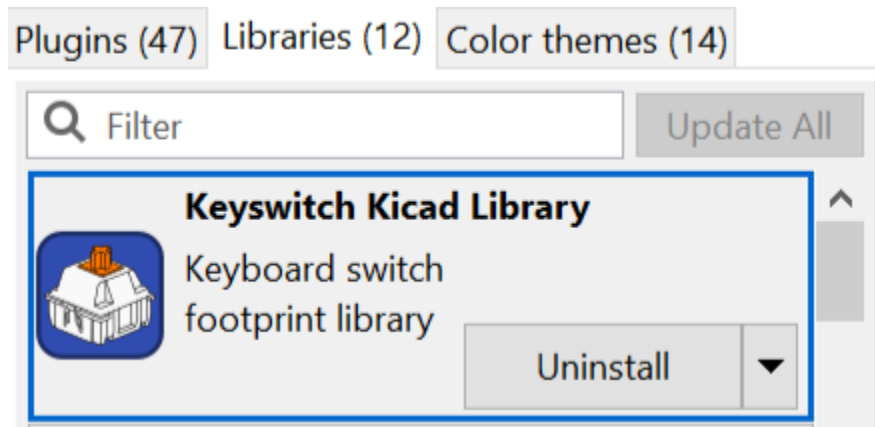
Figure 1: This can be found in the Libraries within the Plugin and Content Manager.

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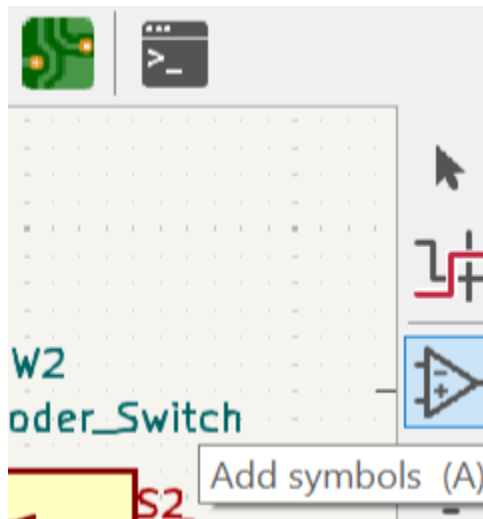

Figure 2: This button will allow you to place parts onto your schematic.

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

4.2   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 defines the copper connections between physical components and the routed traces on a circuit board. To change this, right-click on the rotary encoder and select "Edit with Symbol Editor".
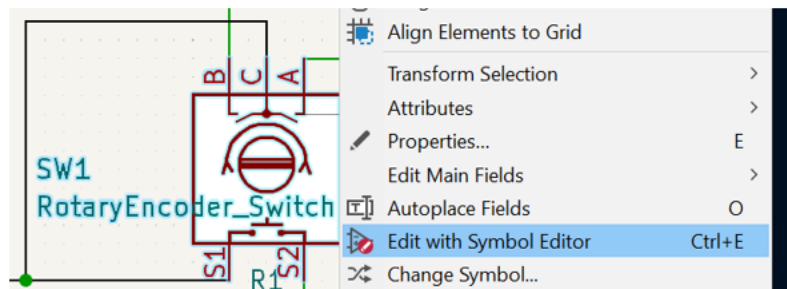

Figure 3: Right click and click "Edit with Symbol Editor".

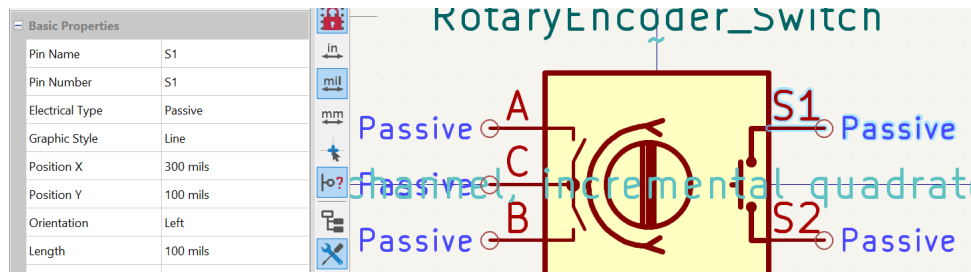Next, swap the pin name and pin number to avoid issues during the tracing part of the PCB design process.

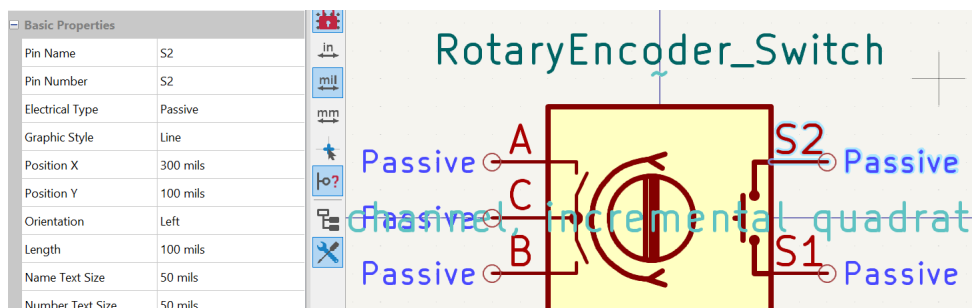Figure 4:   From this Figure above
To the figure below

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

4.3   Final Wiring

Below is the final wiring of what the end result should be. Follow the connections shown in the image below. Click on each connection and match it to its GPIO pin.
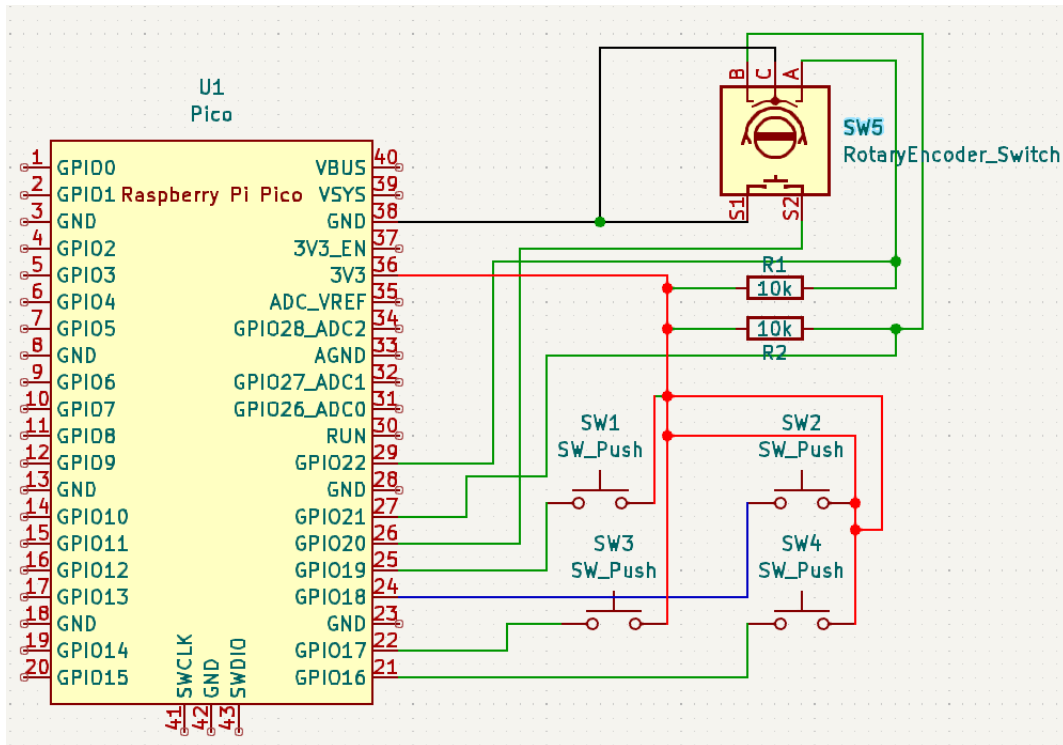

Figure 6: This is how your schematic and wiring should look!

4.4   Assigning Footprints

To assign footprints, simply double-click a part, click within the footprint field, and then click on the small set of books icon located 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.

5 How to use the PCB Editor

Once we have our finalized schematic, we can proceed to the PCB editor by double-clicking the kicad_pcb file in the project files. Upon opening the PCB editor, you will encounter a blank background.
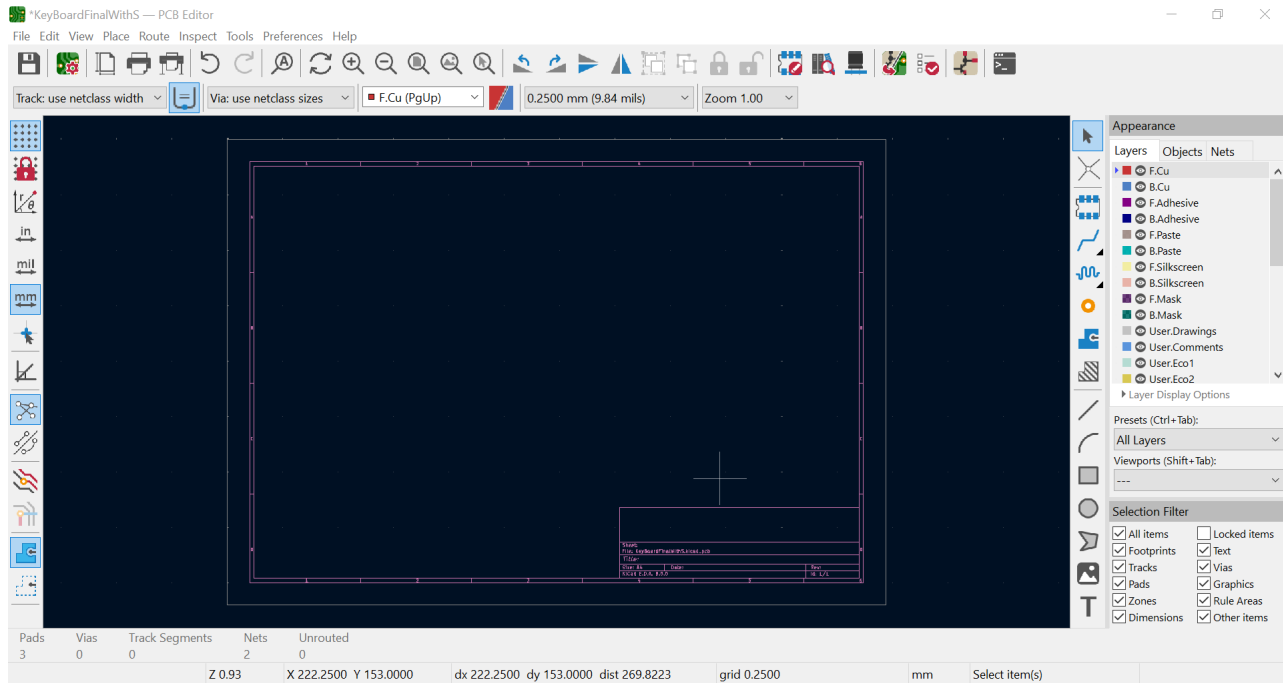

Figure 8: This is how your PCB editor should look when you first open it.

To add our schematic to the PCB editor, we can either click the button located on the top right, situated to the right of the check mark and to the left of the capacitor, or we can simply press F8.
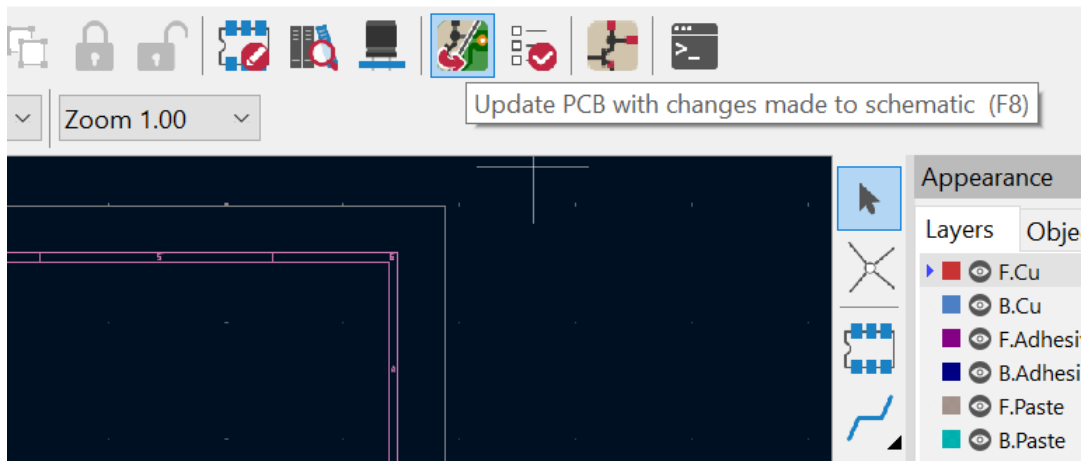

Figure 9: This is where the button is to transfer your schematic.

You'll be prompted with an "Update PCB from Schematic" window. Simply click on "Update PCB" at the bottom. Once you do that, click on "Close", and all your components should be attached to your mouse.

5.1 Arrangement

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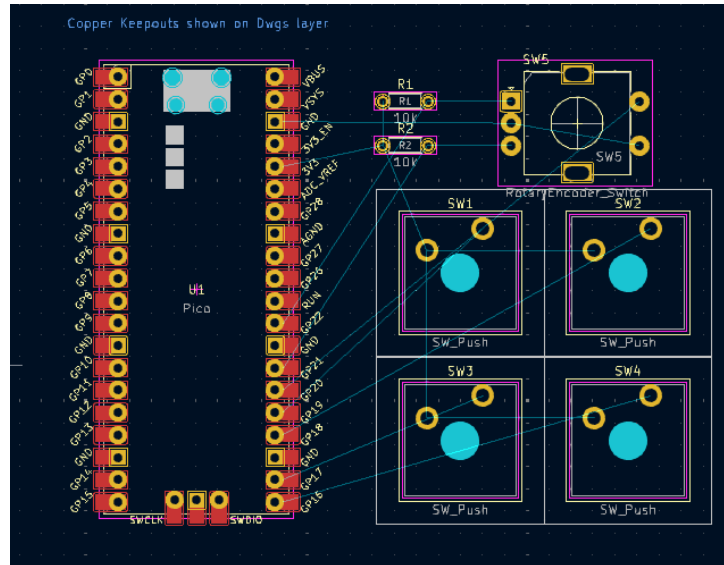

Figure 10: Here is how your PCB Editor should look.

5.2 Edge Cuts

Edge Cuts enable us to define the shape of our PCB. Within the "Appearance and Layers" section on the right side of our PCB editor, locate the "Edge.Cuts" layer. Click on it, then use the "Draw Rectangle" button to draw a rectangle around the perimeter of the PCB.
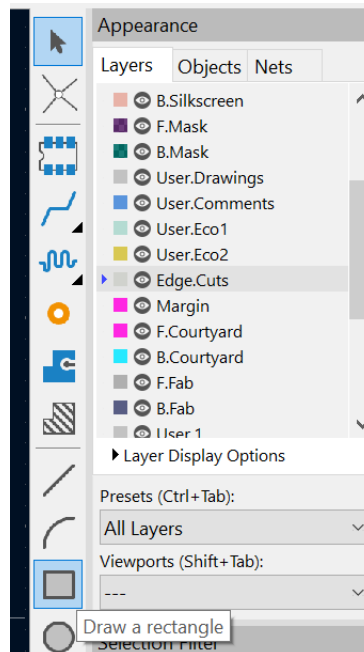

Figure 11: Here is where the Edge.Cuts is located in the Layers tab under Appearance.

5.3 Tracing

To start tracing, make sure you're on the Front Copper layer or "F.Cu". Then, hover your mouse over any of the yellow pads that have lines connecting them and press "X".
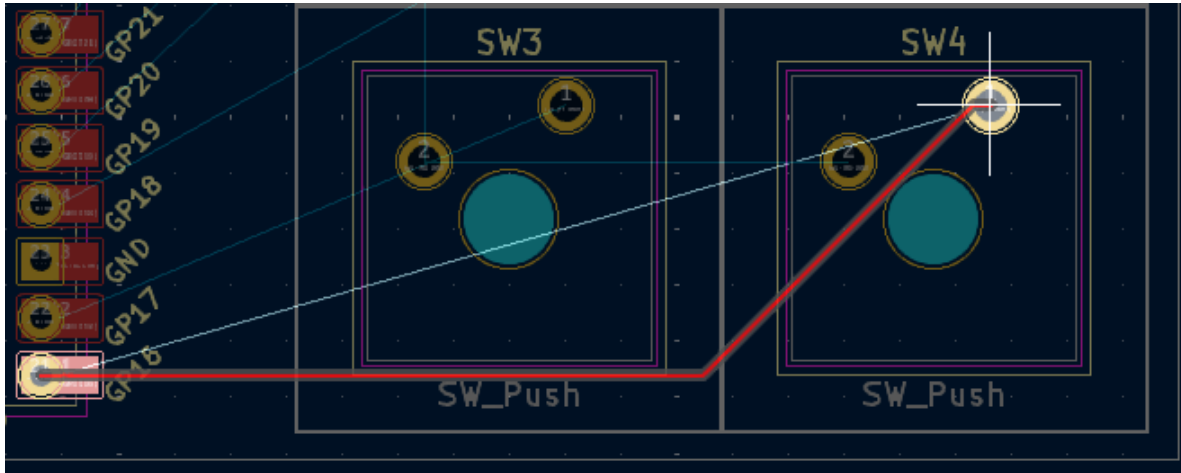

Figure 12: Here we can see GPIO18 being connected to Pin 1 on Switch 4.

If you find there isn't enough room to route your traces, you can switch to the back of the PCB by changing your layer to "B.Cu" or by pressing the minus key to toggle between the layers. Remember, your routing may differ from others', but as long as the connections are touching, you're good to go.
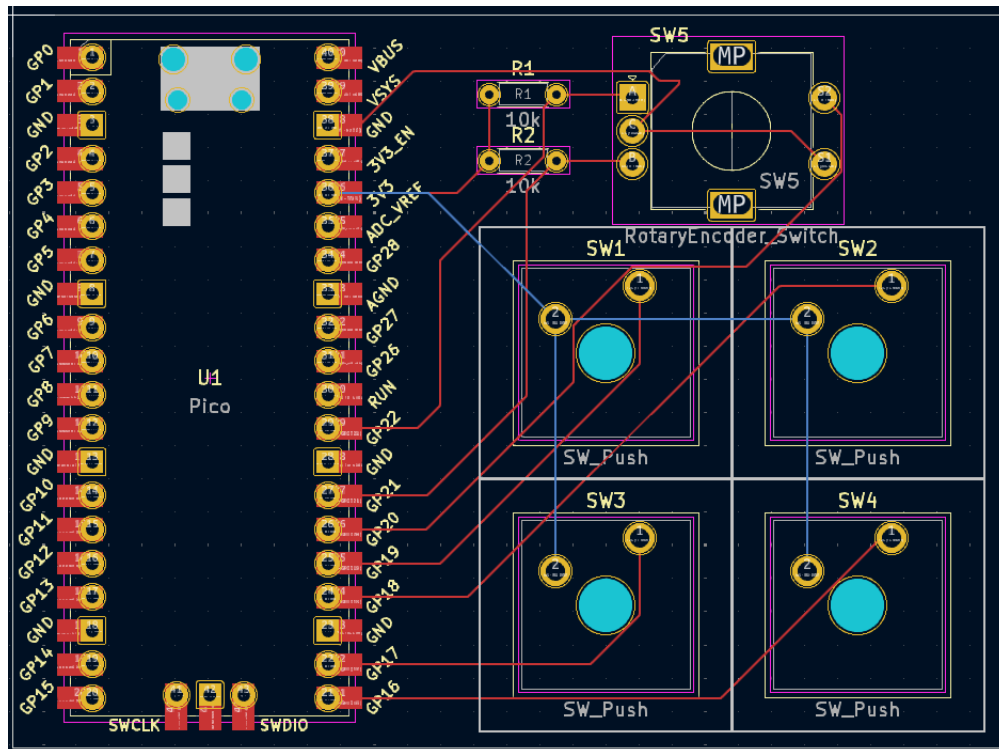

Figure 13: Here is an example of what your PCB could look like.

5.4 Ground Plane

Adding a ground plane helps reduce electrical noise, interference through ground loops, and prevents crosstalk between adjacent circuit traces. To add a ground plane in KiCAD, locate the "Add a filled zone" button on the right side of the screen, situated under the yellow circle.
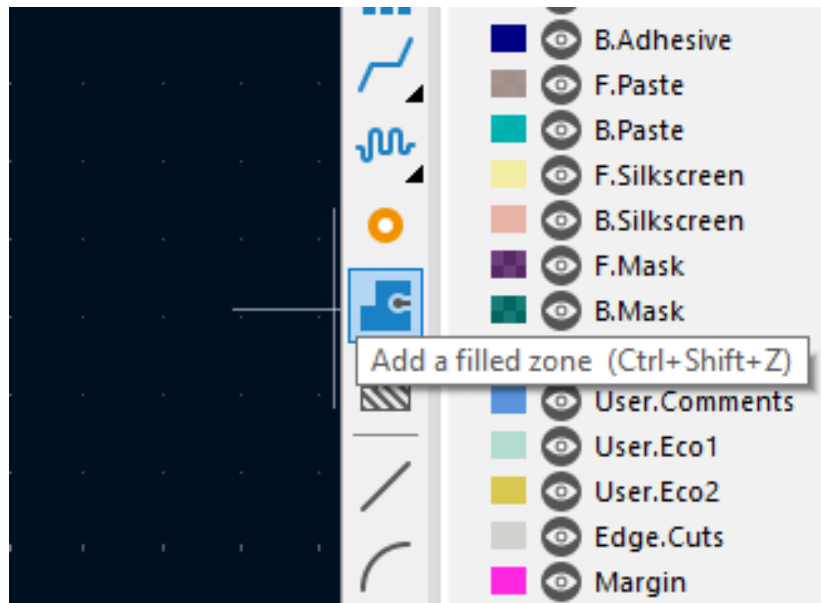

Figure 14: Here is what the button looks like.

Once selected, click on the top left of your edge cut or the outline of your PCB. If you routed traces on the back of the PCB, ensure you select "B.Cu" as well within the popup. After that, click "OK" and proceed to click on each of the remaining corners of the PCB. Once you click the corner you started with, you should see an outline appear around your PCB.
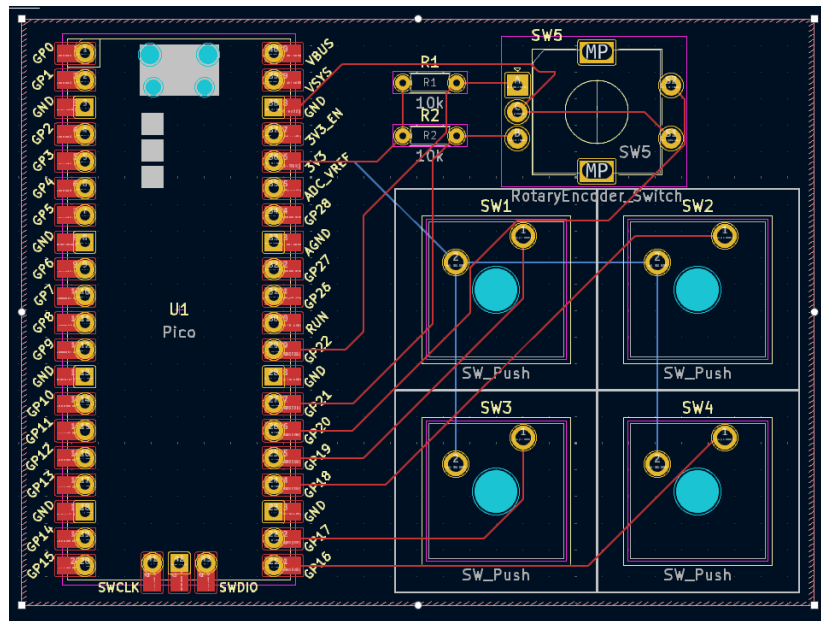

Figure 15: Here is a PCB an unfilled zone.

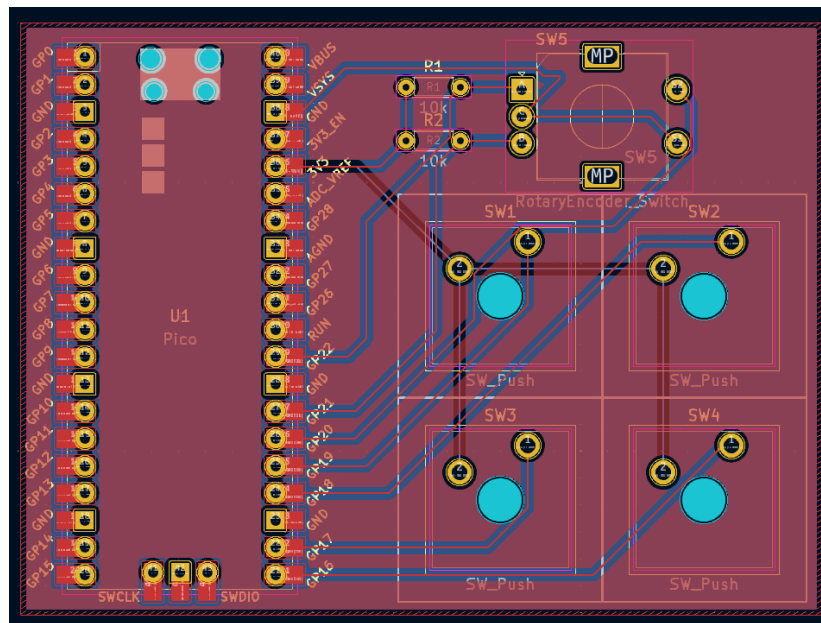All that is left to do it press "B", and you should see your PCB automatically fill in.

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

5.5   3D Models

KiCAD offers a useful feature: the ability to view a 3D rendering of your PCB. However, we need to ensure that we're using the correct 3D model for each part.

Let's start with the Raspberry Pi Pico. Double-click on it and navigate to the "3D Model" tab. From there, locate the appropriate 3D model.
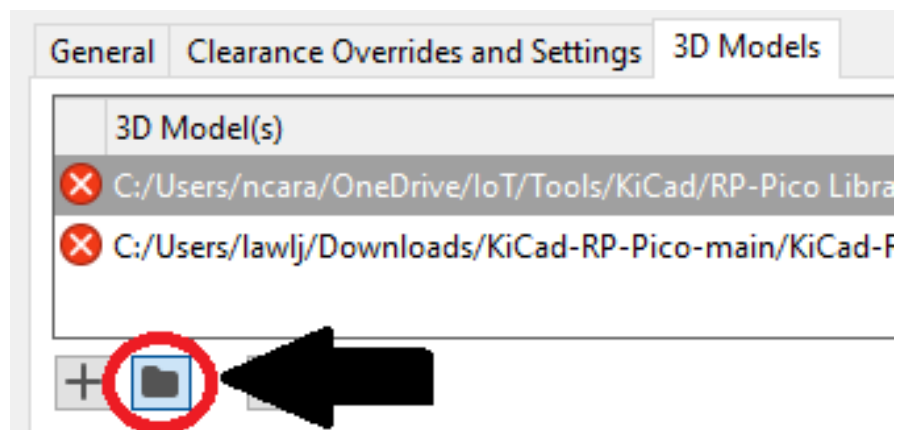

Figure 17: Click on this to enter the file location.

Fortunately, KiCAD offers the flexibility to switch between different available paths. You'll need to locate the folder you're working within, as it contains the 3D models for the Pico and the Rotary Encoder.
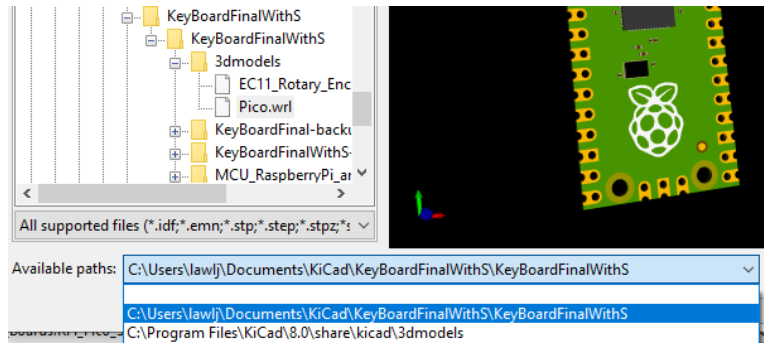

Figure 18: Here is what your file location should look like.

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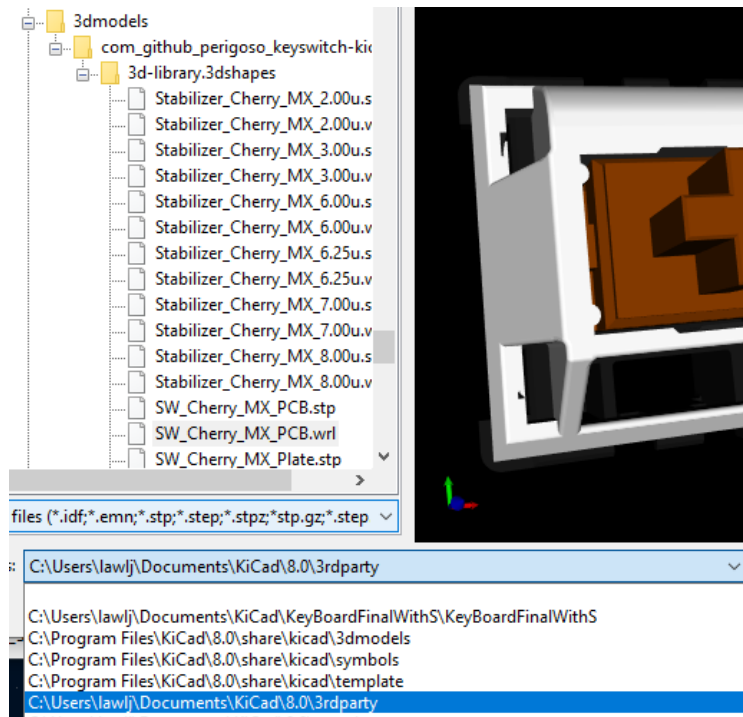

Figure 19: File path and part name

One thing to note about the switches is that they will need an x offset of -2.5 and a y offset of -5.0 to position them correctly on the PCB. Similarly, the rotary encoder requires a 90° rotation on the z-axis and an x offset of 7.5 and a y offset of -2.5 for proper placement.

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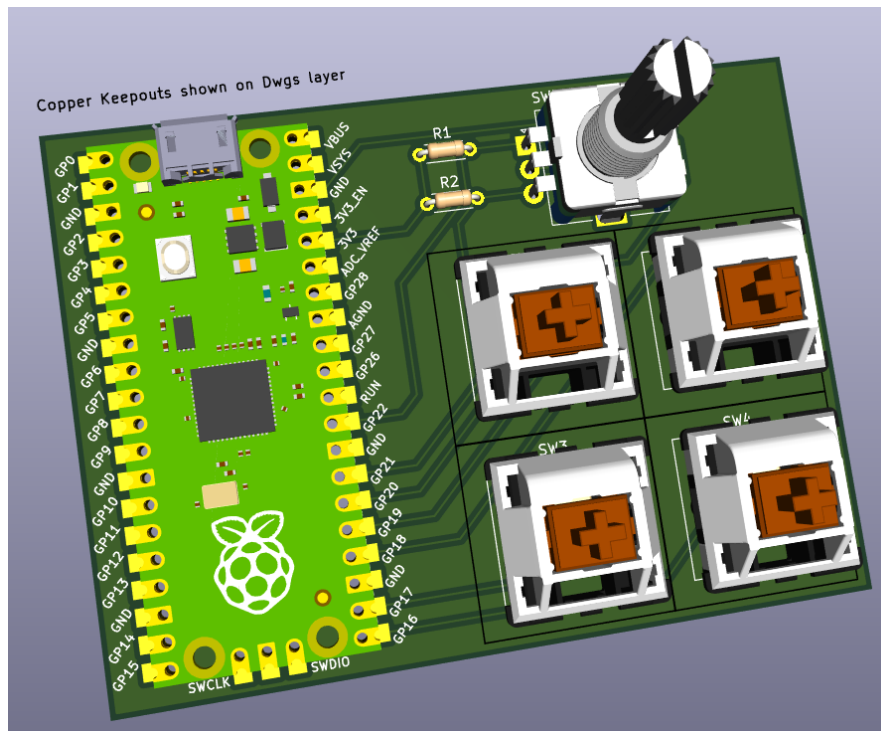

Figure 20: Here you can see a 3D model of your PCB

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