RoboJackets Electrical Training Week 4 Lab Guide

Kyle Nguyen

August 23, 2024
v1.0

Contents

1   Background

This week, we are addressing how to create the Board Layout design of a PCB. There are many specific details to the art of designing a PCB, but our main intention is to introduce you to the basics of it all. Every time that you want to make a PCB for attending your specific application, you need to go through some steps and that's not different in RoboJackets. Every team has several different boards that support either their main robots or parallel projects that are relevant for the team. On this lab, we will design the PCB Layout of the Firmware Training Board. This is the same board which you all have made the schematic last week.

2   Objective

2.1   Board dimensions

The board size depends a lot on the application and on the constraints set by the components that will go on the board. For example, smaller boards are preferable if you want to put your board inside a tight system like a robot (and in most cases actually), however if the end-application don't have many physical constraints, you can make your board bigger and thus make it easier to routing traces on it and soldering the components after the PCB is manufactured.

2.2   Optimize User Experience

In the end, you want your board to look good and to be informative enough for the end-user.

2.2.1   Silkscreen

Use the Silkscreen layers to put information for people assembling or using your board. In general, people tend to put less information than it would be helpful, however, don't put too much unnecessary silkscreen to a point of polluting the board view. You can also put graphics and silly messages to people on the silkscreen (sorry Jinhee, but you need to get out of bed).

2.2.2   Space between components

Especially if your board is going to be soldered, remember to put some space between components so that if it needs to be assembled or debugged, the user can easily take out or put components.

3   Relevant Information

3.1   Datasheet!

You thought that you only would need to analyze datasheets while doing the schematics and choosing the components for your board? Nope! The datasheet has very useful information on how to do the layout. It will suggest the positioning of certain supportive discrete components (like resistors and capacitors) and how should the polygons and traces around the main component look like. In fact, it will commonly show you an example of a working Board Layout like in Figure 1.
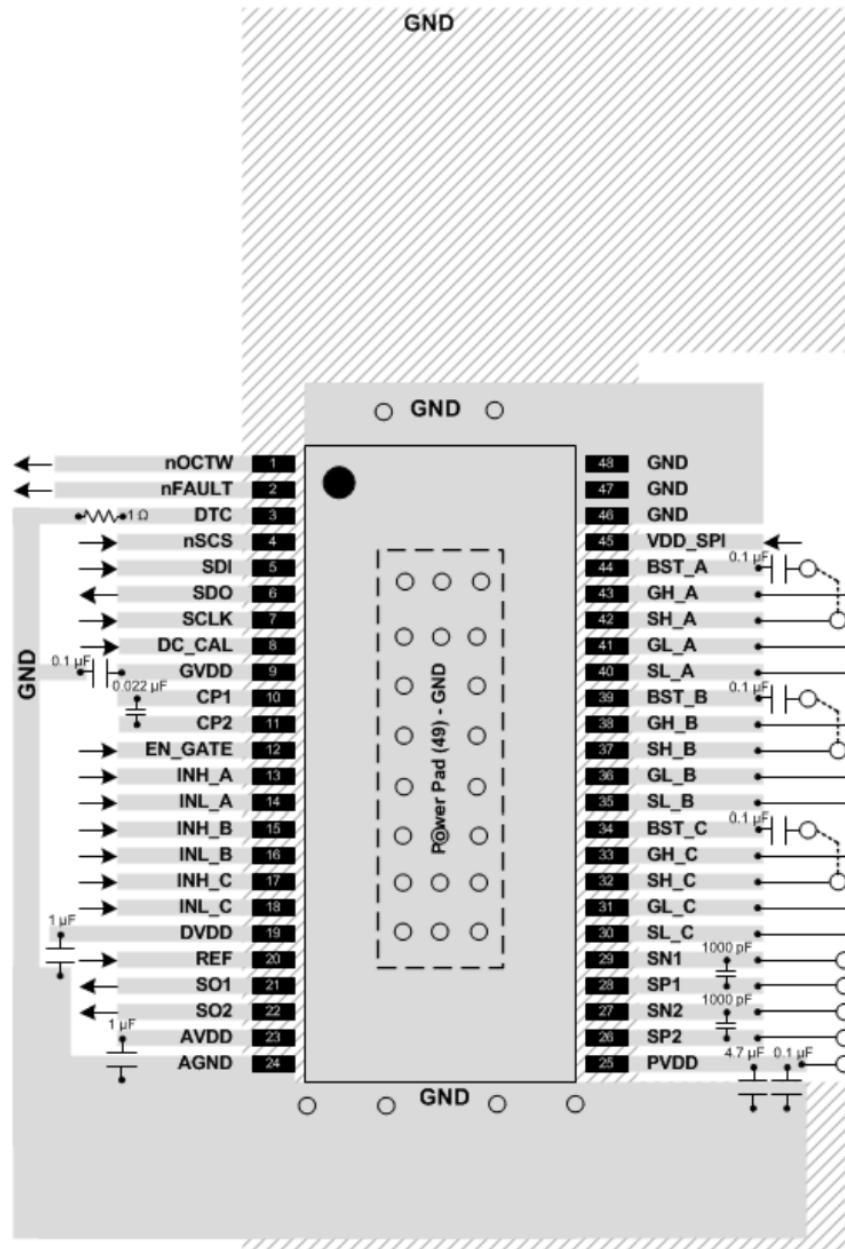
10.2 Layout Example

Figure 1: Layout example from a component's (DRV8303) datasheet. Note it suggests the positing of capacitors and the width of certain traces.

3.2 Terminology

Here are some terms you should be comfortable with:

- Layers: Different categories for elements in the board. The KiCAD PCB editor documentation has some details on some of the important KiCad layers: https://docs.kicad.org/7.0/en/pcbnew/pcbnew.html

- Trace: The filament of copper that connects pins around your board.

- Via: Copper platted hole that goes from top to bottom layers and passes by all the other layers.

- Polygon: Region of board designated to be entirely filled with copper and thus can be used to connect pins.

Color	Layer Name	Layer Purpose
	F.Cu	Top/front layer of copper
	B.Cu	Bottom/back layer of copper
	Edge.Cuts	Outline of the board
	F.Silkscreen	Silkscreen for top
	B.Silkscreen	Silkscreen for bottom
	F.Fab	Top/front documentation layer (just for reference)

Figure 2: Quick explanation of layers you're most likely to use.

4 Guided Lab

4.1 Setup

Open the KiCAD schematic from the last lab. If you were unable to come to the last lab(s), download the ElectricalTrainingLab.kicad_sch file in the Week 3 folder in the electrical-training repository in Github for you to download. Copy and paste the contents of the schematic file into your own project. If you don't have a project open because you don't like seeing me during these electrical training meetings, make one by going to File at the top left and clicking New Project.

Open your footprint library and make sure the following footprints are in the library:

- Teensy 4.1 (Microcontroller)

- Cherry MX Black Switches (MX1A-11NN)

- Motor Driver (A4950ELJTR-T)

- Voltage Regulator (TSR 1-2450)

If the any of these are not in your footprint library, please talk to an instructor.

4.2 Assign Footprints

- With your schematic open, go to **Tools ¿ Assign Footprints**.

- There may be footprints already assigned to certain symbols, but delete those for now. We want to make sure that the footprint that we want is the one we actually want (just click on it and hit delete).

- In the search bar, type in **0805 capacitor hand solder** and select the option that pops up. Assign this footprint for all of the capacitors in the list. 0805 is the size of the capacitor. For more information on these types of footprints, click on this link.

- In the search bar, type in **805 resistor hand solder** and assign this footprint to all of our resistors.

- For the battery connector, search **XT60 1x02 Vertical** and assign the female option to this symbol.

- For LeftMDConn and RightMDConn, search **Conn Phoenix 1x02 5.00mm** and select the horizontal version without the threaded flanges.

- For the rest of the parts, search up the footprint names and assign them to their respective symbols.

Your footprint assignment screen should look something like this now:
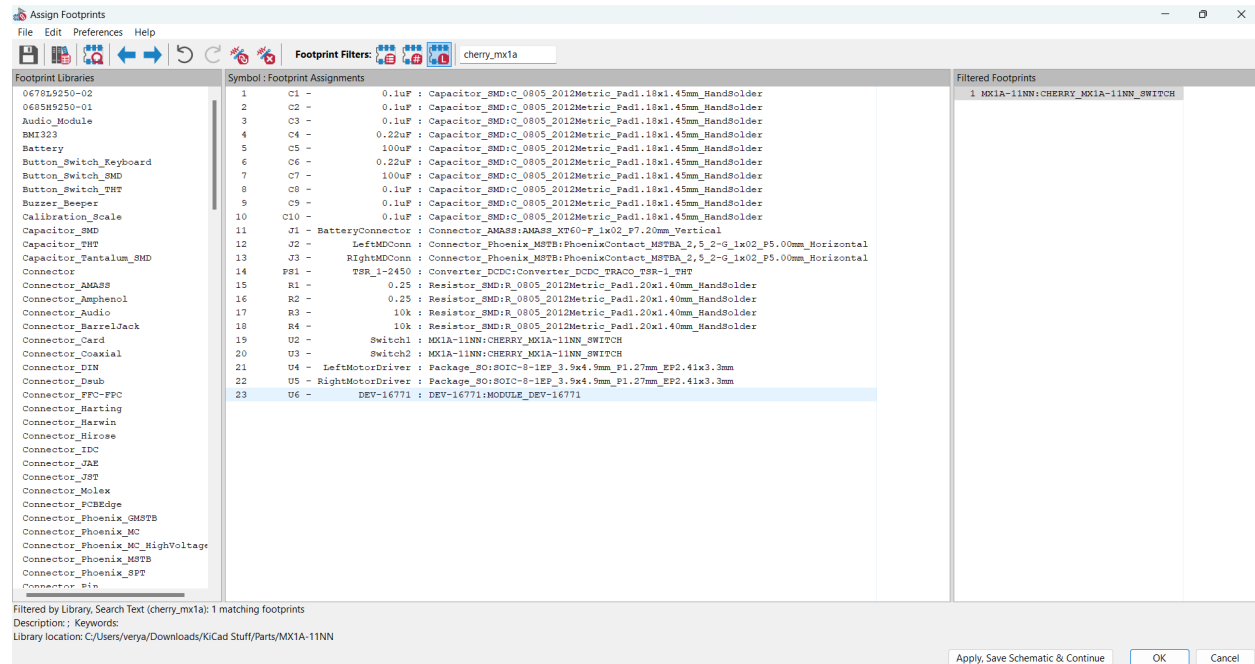

Figure 3: Footprint assignment in KiCAD with all parts filled out for a specific PCB.

4.3   Update PCB from Schematic

- In your schematic window, go to **Tools** at the top and then click **Update PCB from Schematic**.

- You will be redirected to the PCB Editor window. Click **Update PCB** and all of your footprints should appear.

4.4   Setting the PCB Bounds

- Select **Edge Cuts** in the right panel under **Layers**.

- Select **Draw Rectangle** from the panel on the right side (not under Layers).

- Make a 72mm x 70mm border. Normally, you would check with your mechanical member about the size of your PCB, but since I'm your instructor, you will make it 72x70 mm.

4.5   Footprint Placement

Footprint placement is purely subjective. As the designer, you are allowed to place parts wherever you want. However, here are some suggestions:

- Place the Teensy so that the USB side is close to the edge of the board so it's easy to plug stuff into it.

- Keep the right motor driver components on the right side, and keep the left motor driver components on the left side.

- Keep related parts next to each other. For example, if a specific resistor belongs with a switch, keep those two footprints close together.

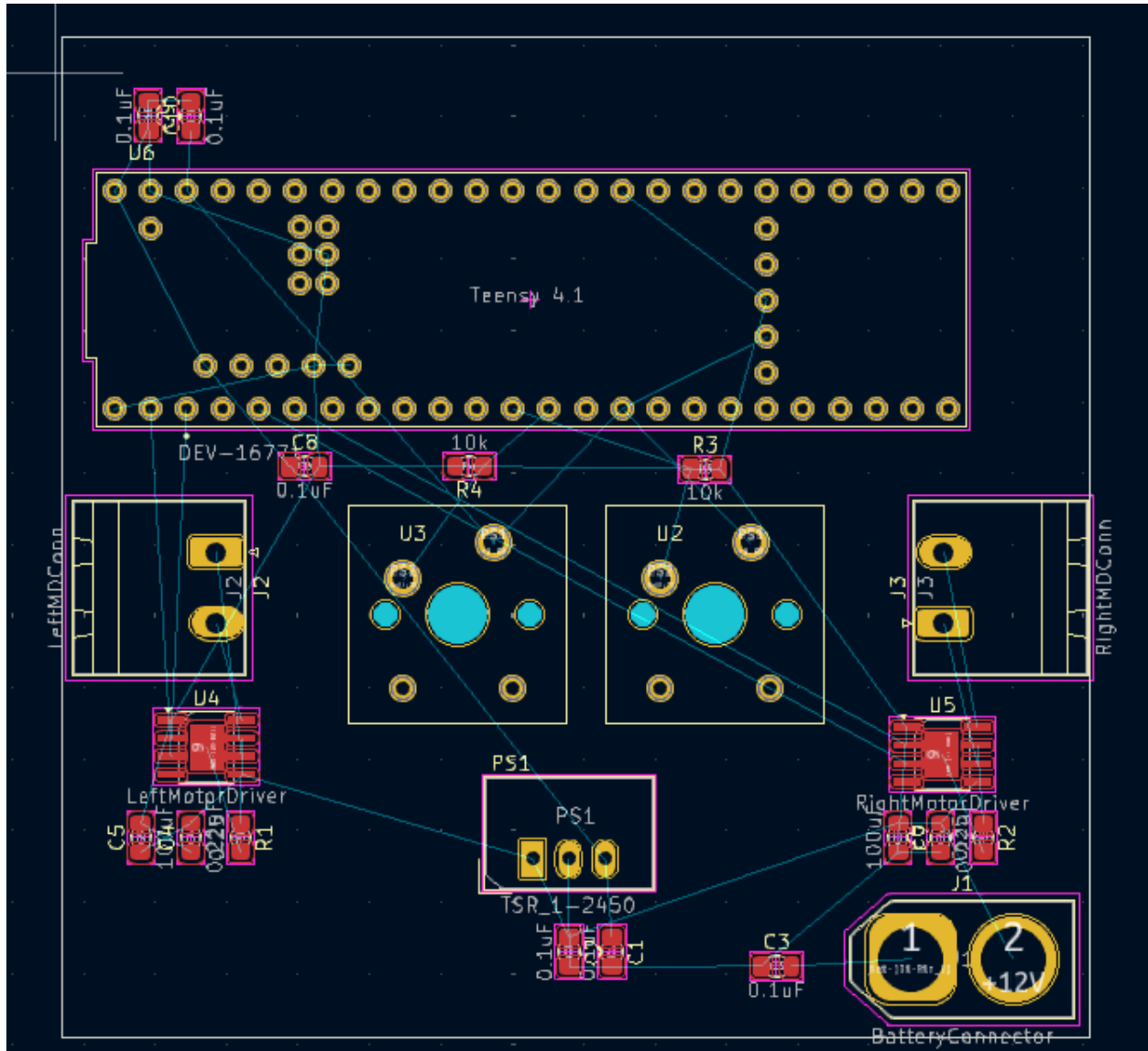Here is an example of how you could place the footprints:

Figure 4: A possible layout for a specific PCB in KiCAD. Please note that there could be a better way to do this.

4.6 Wiring

- **DO NOT WIRE GROUND**. We will take care of that later.

- Press X on a pad to start drawing a wire. The blue lines indicate where you need to connect that wire to.

- If you find yourself trying to cross another wire, press V to place a via. The wire should change colors (either from red to blue or blue to red, depending on what layer you were on before) and you should be able to cross that wire.

- Wire everything that you need to until you have only grounds left.

- You may need to rotate or move some parts so wiring is easier.

- You may also need to change pin assignments on your Teensy if you want to make wiring easier.

4.7 Ground Layer

- To place the ground layer, select **F.Cu** in the **Layers** menu on the right, and then click **Add Filled Zone** in the panel on the right (not under Layers tab).

- Click on a corner of your PCB, and a menu should pop up.

- Under **Layer**, select both **F.Cu** and **B.Cu**. Under **Net**, select **GND**. Click **OK** to continue.

- Click the rest of your PCB corners to create the borders for the ground layer.

- Click **B** to fill in the zone.

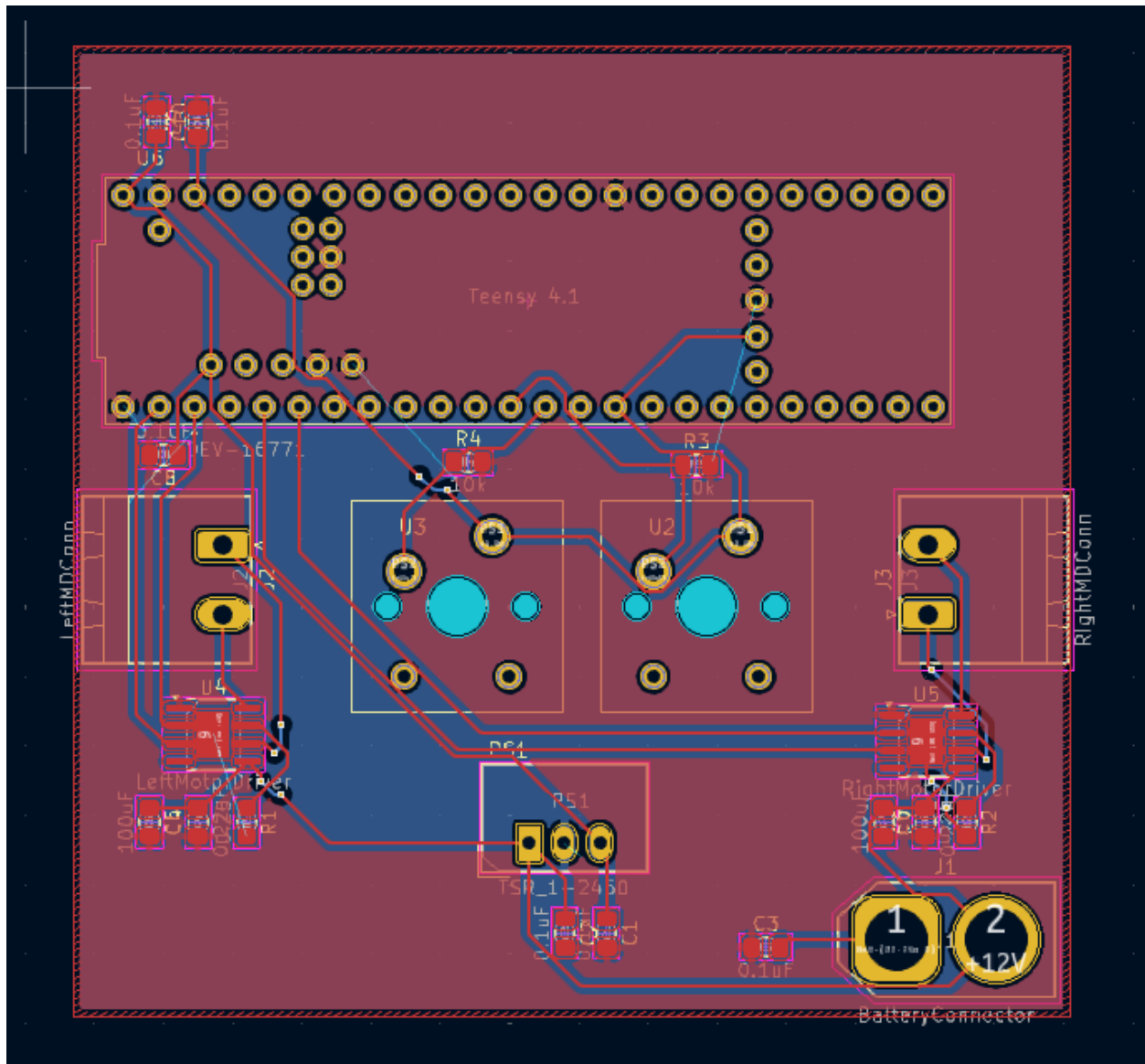Your PCB should look similar to this now:

Figure 5: A complete wiring for a specific PCB in KiCAD. Note that the wiring and placement does not have to be the same as this; in fact the wiring could have been done much more efficiently if I wasn't lazy.

5 Troubleshooting

To make sure that your board layout design abides by the Design Rules that you have established in the beginning of this guide, press the DRC tool (looks like a checklist) in the top bar and a screen should appear. After you click **Run DRC**, it should describe the infractions you have committed. Thus, you can press on the errors and warnings and the screen will take you to them. You can ignore some errors, but some you cannot. If you are unsure about an error, ask your instructor.

If you are struggling, here are some resources that may help you.

- KiCAD Getting Started Guide - Includes how to use the schematic editor

- Git Guide

- RW PCB Tutorial