

RoboJackets Electrical Training Week 4 Lab Guide

Jackie Mac Hale

October 17, 2023
v2.0

Contents

1	Background	2
2	Objective	2
2.1	Board dimensions	2
2.2	Optimize User Experience	2
3	Relevant Information	2
3.1	Datasheet!	2
3.2	Terminology	3
4	Guided Lab	4
4.1	Update your footprint library	4
4.2	Open the file	4
4.3	Set the Design Rules	5
4.4	Shape the board	5
4.5	Positioning the Components	6
4.6	Ground Plane	8
4.7	Routing	8
4.8	Silkscreen	8
4.9	Check how your PCB looks	9
5	Troubleshooting	10

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.

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

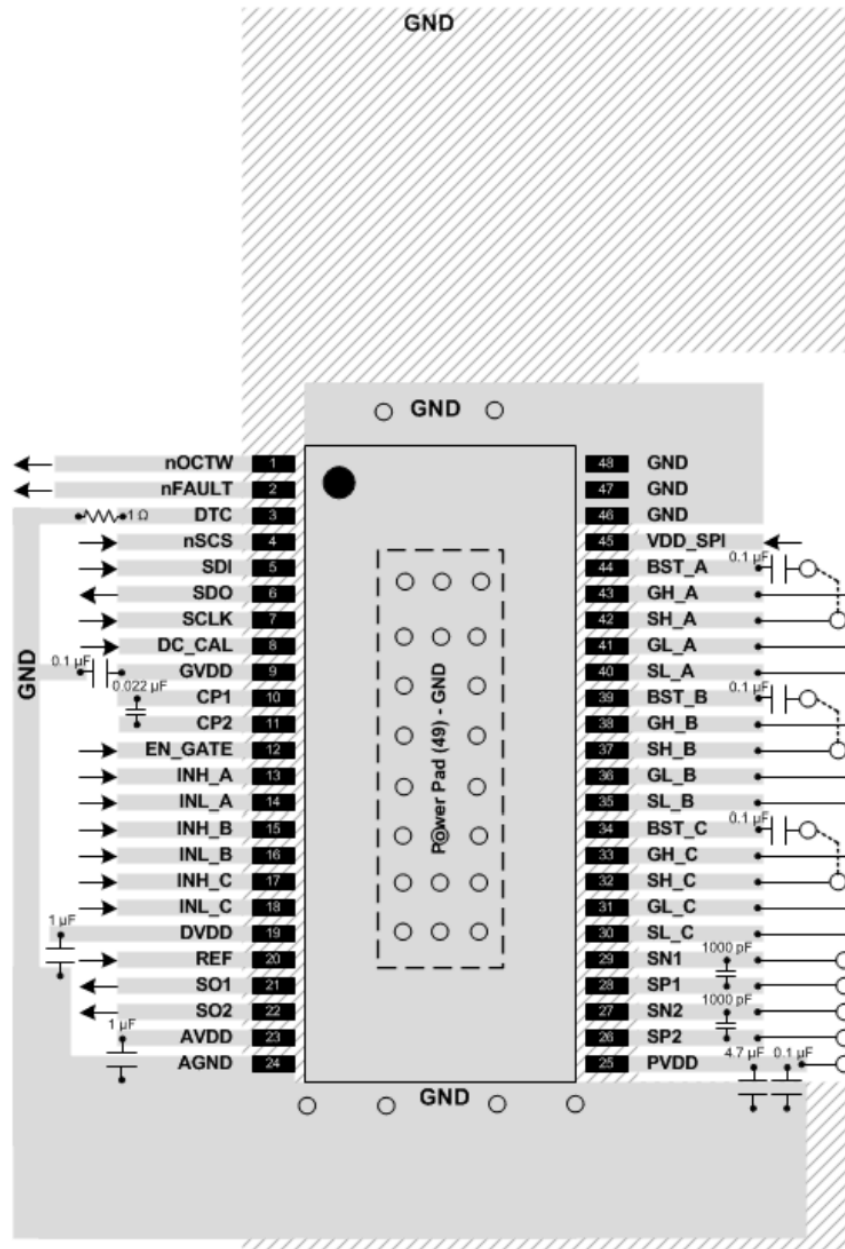


Figure 1: Layout example from a component's (DRV8303) datasheet. Note it suggests the positing of capacitors and the width of certain traces.

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 plated 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 Update your footprint library

In the KiCad Project Manager Window, go to Preferences > Manage Footprint Libraries to add the necessary footprints for this library. Make sure the Global Libraries tab is selected and click on the folder button to add the RoboJackets.pretty and SparkFun-Connector.pretty folders which can be found in electrical-training > kicad-libraries > footprints.

4.2 Open the file

4.2.1 Find Week 4 project

In your KiCad project manager window, open the “week-4-schematic.kicad_pro” which can be found in the labs folder of the electrical-training repository. Then, open the schematic editor.

4.2.2 Switch to Board Layout view

Click the “Open PCB in board editor” button to create a board layout file. Click yes to create the board file.

It's recommended to switch your grid style to “small crosses” in Preferences > Preferences > Display Options under PCB Editor since lines look really dense in the PCB editor.

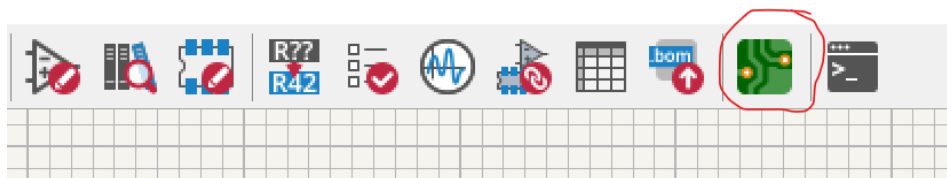


Figure 3: This is what the button looks like!

4.2.3 Page Settings

Change the size of the paper you're drawing to US Letter in File > Page Settings. Add today's date, assign a revision number of 1.0, and write a descriptive title for the board you're creating.

4.2.4 Add footprints

Click the update PCB button in the top bar to add the footprints to your editor. Make sure to select the second and third options to import all of the footprints.



Figure 4: This is the button you should press!

4.3 Set the Design Rules

You will need to set the constraints for the board layout you want to create. Those constraints are commonly set due to limitations of the PCB manufacturers. For instance, some manufacturers can drill 0.1 mm holes on your PCB, but some others are only able to do holes bigger than 0.3 mm.

So, for this lab, you should load the design rules of JLCPCB which you can find in the kicad-libraries folder of the electrical-training repository that you should have cloned previously (remember to pull to get the most recent files!).

To load it, go to File > Board Settings. In the window that pops up, go to Custom Rules under Design Rules and copy + paste the JLCPCB design rules text file into the DRC rules text input section with line numbers.

4.4 Shape the board

First, adjust the grid size to 1 mm in the top bar.

Since this board will be attached to the top of an Arduino Uno by connecting to all of its exposed pins, our main constraint is going to be those predefined pins. Then, we will preferably want to make the board smaller or the same size as the Arduino Uno board like the example in Figure 5. For that, you can also use the Arc tool to shape the corners.

Click on the Edge.Cuts layer and draw the board outline with any of the drawing tools in the right menu.



Figure 5: This would be a good board shape.

4.5 Positioning the Components

You have a lot of space for positioning your components and there's no specific component that has strict positioning constraints so just try making the least amount of crossings between airwires. For that, select a footprint and then drag it to where you want to place it. You can rotate by pressing the 'R' button.

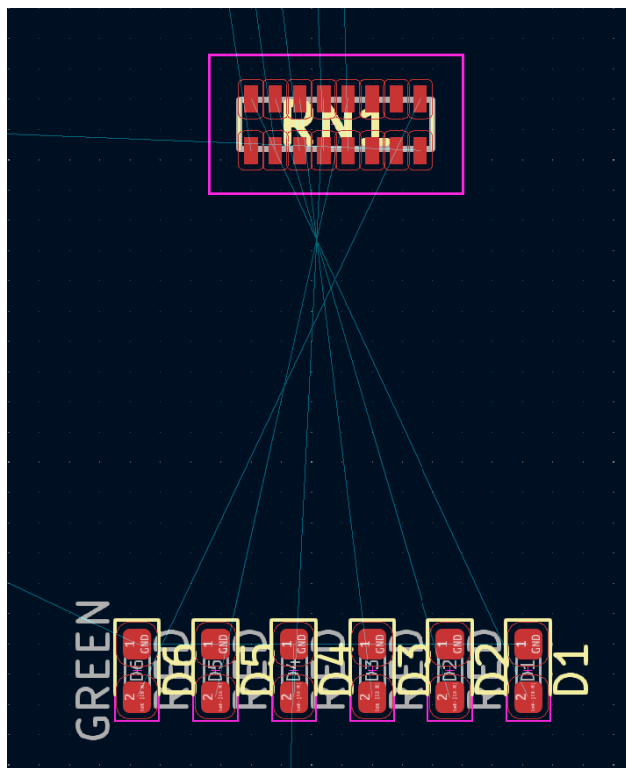


Figure 6: BAD - Many crossings.

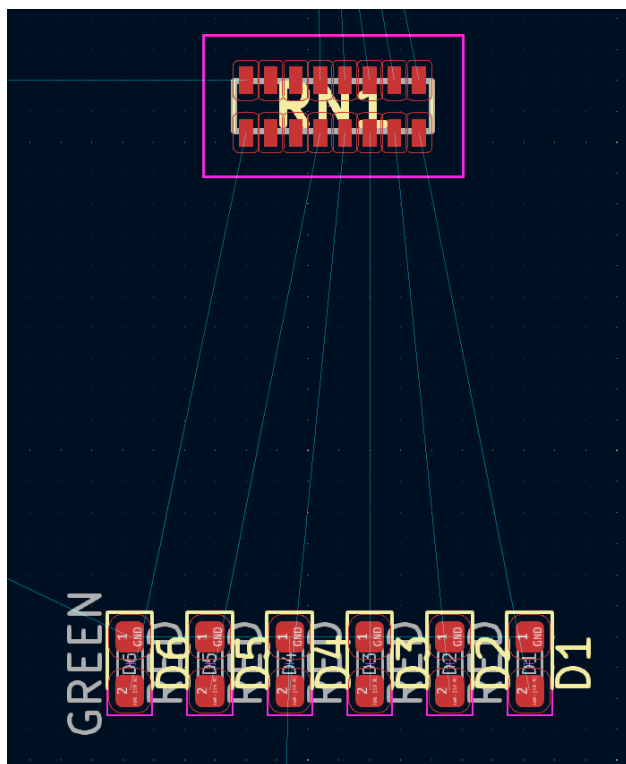


Figure 7: GOOD - No crossings.

4.6 Ground Plane

A polygon plane is important for minimizing the number of traces used for ground and for giving an easy return path for the current that comes out from your circuitry. Doing it is fairly simple:

- First, select the layer you want the plane on.
- Then, click on the “Add a filled zone” button in the right menu. Click on a point in the layout editing area you want to start creating a plane from to open a menu. Select the signal you want to make a plane for. Hit OK and draw the polygon to create a plane. Double click for the last line to finish the polygon.
- To fill the plane, you can go to Edit > Fill All Zones.

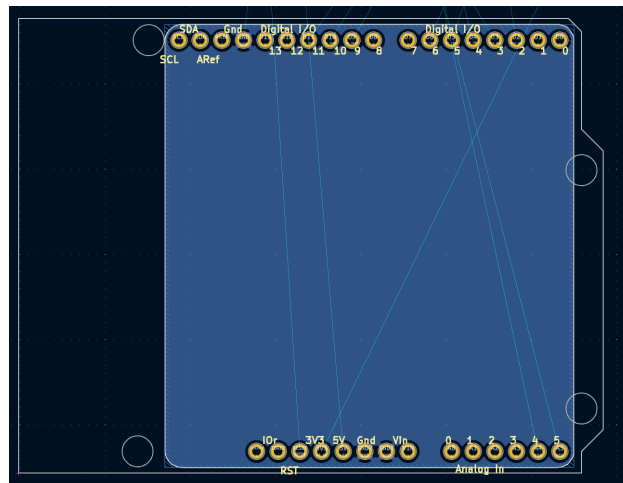


Figure 8: A ground plane example.

4.7 Routing

Use the “Route tracks” tool to connect all of the airwires. If you need to cross any two traces, you can route a trace underneath the other by putting a via, going to the bottom plane and then, putting another via for the trace to return to the top layer.

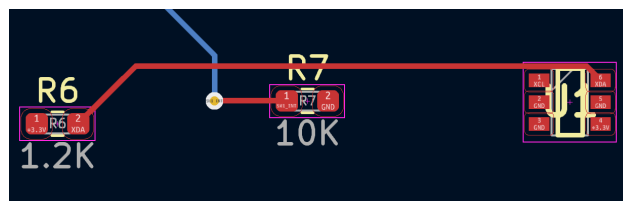


Figure 9: You can route the same signal on different layers.

4.8 Silkscreen

Add text on the F.Silkscreen layer with the Tex tool to label the LEDs as well as the buttons and the 5V on the 8-pin connector.

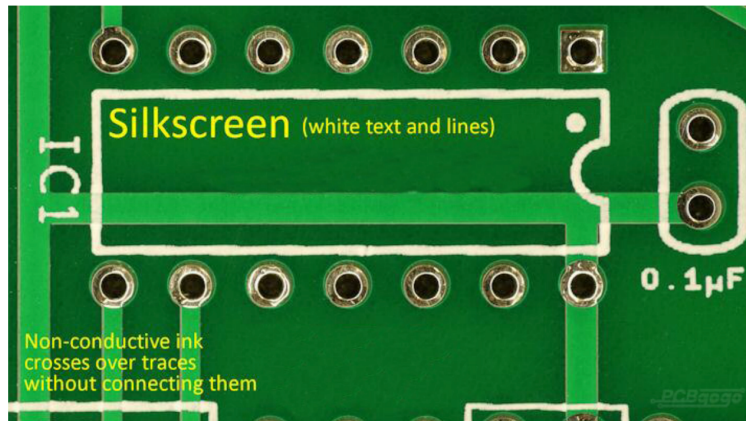


Figure 10: Silkscreen on a PCB.

4.9 Check how your PCB looks

After you've finished laying out your PCB, you can generate Gerber files for it to see if everything looks good. You should check if there's no silkscreen names or labels under components and see if everything seems informative enough.

To do this, go to File > Plot. Specify/create an output directory so that the gerber files can be contained in its directory instead of the just the project directory which is the default. Click "Plot" to generate the files.

Open the Gerber Viewer from the Project Manager window and open the generated files in File > Open Autodetected Files. Select all of the files except the .gbrjob file and open them. Your board layout should show up.

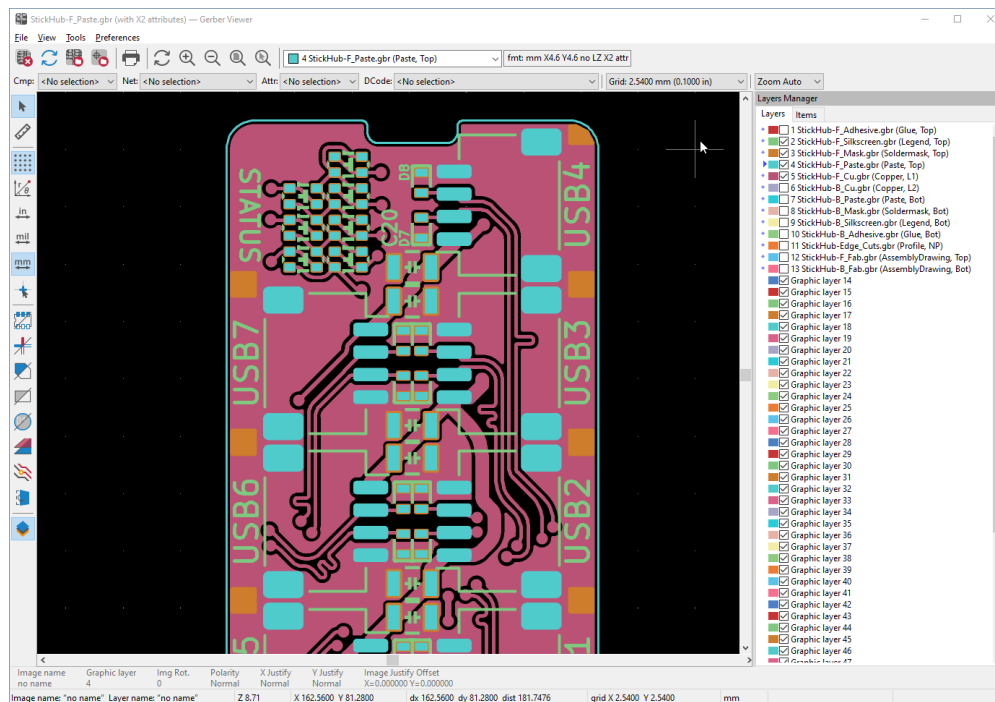


Figure 11: Example of a finished board in the Gerber Viewer.

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. Make sure you solve every complaint until you don’t see any warnings/errors in the DRC tool.

- [KiCAD Getting Started Guide](#) - Includes how to use the schematic editor
- [Git Guide](#)