Ordering Simple 2-Layer KiCad Designs From JLCPCB - UMD Balloon Payload Program

Introduction

These instructions have been written for members of the UMD Balloon Payload Program who want to order relatively simple printed circuit boards design in KiCad. Readers should have already designed a PCB and be ready to order. The instructions below will guide readers through checking the design for design rule violations, generating fabrication outputs, and finally ordering the design.

Technical Background

Readers should be familiar with the PCB design process and general construction of PCBs. They should have already designed a PCB and should be ready to validate it and order it. More information about the design and construction of PCBs can be found at the blog post below.

What is a Printed Circuit Board? Make Circuits by Connecting Components

A printed circuit board is an electrical circuit whose components and conductors are contained within a mech...

https://resources.altium.com/p/what-is-a-pcb

Materials

- A Computer
- Access to the internet
- A fully designed and routed PCB in KiCad version 7 or higher
- A payment method (credit/debit card)

Procedure

Design Rules Setup

1. Begin by opening your KiCad project and entering the PCB editor.

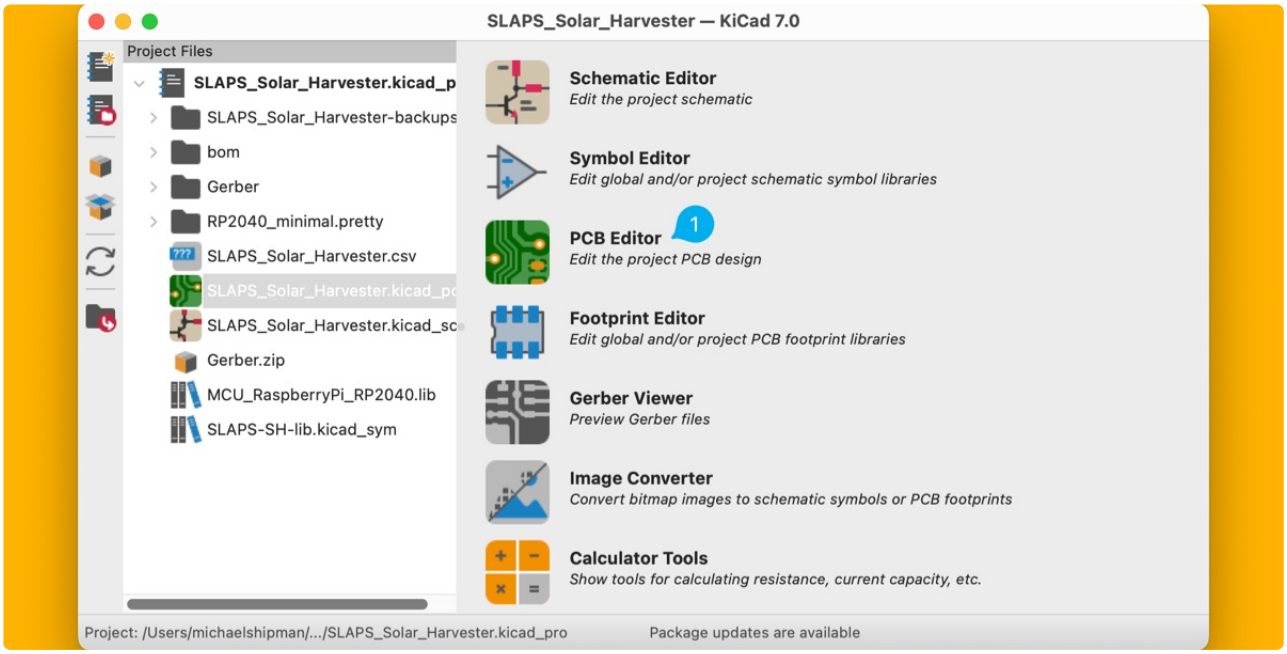

Figure 1

2. Open the menu `File > Board Setup`

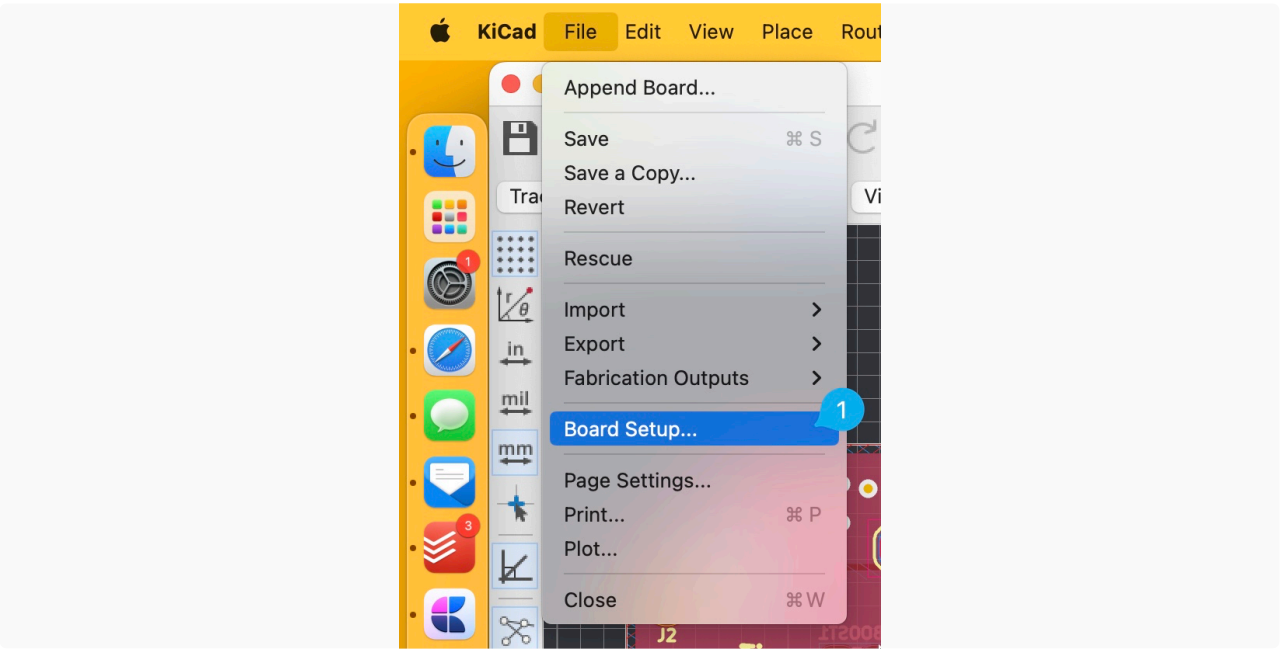

Figure 2

3. Open the `Constraints` menu

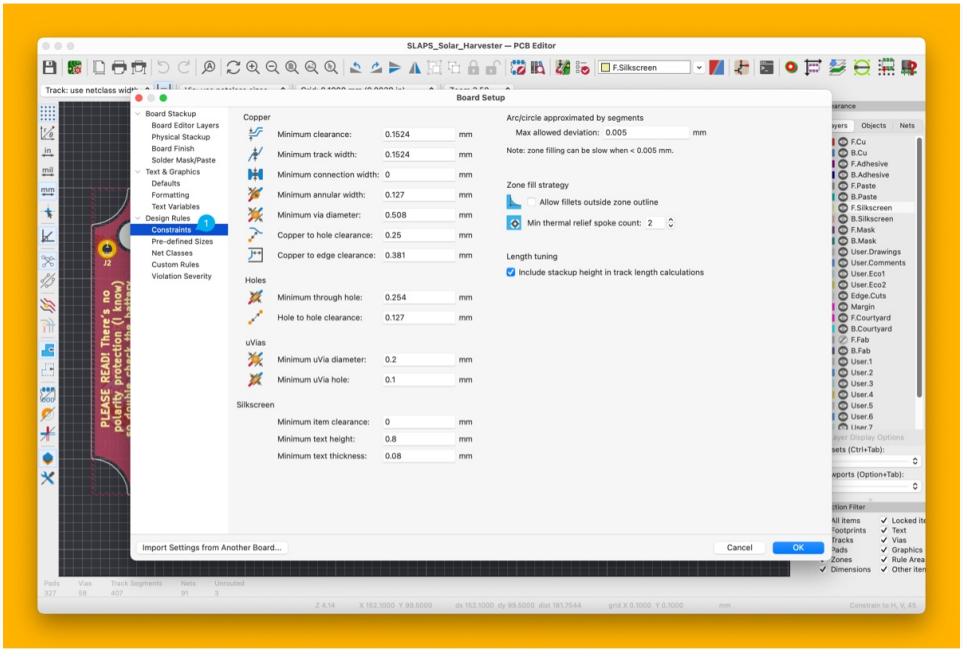

Figure 3: The constraints menu containing filler data

4. Enter the design constraints found at https://jlcpcb.com/capabilities/pcb-capabilities

> KiCad parameter name: JLC parameter name

1. Minimum clearance: Minimum trace width and spacing > Min. spacing
2. Minimum track width: Minimum trace width and spacing > Min. Trace width
3. Minimum connection width: Minimum trace width and spacing > Min. Trace width
4. Minimum annular width: Minimum Annular Ring > Minimum annular ring
5. Minimum via diameter: Drill/Hole size > Min. Via hole size/diameter
6. Copper to hole clearance: Minimum clearance > PTH to Track
7. Copper to edge clearance: Board Outlines > Trace to Outline(Routed)
8. Minimum through hole: Drill/Hole size > Drill Hole Size
9. Hole to hole clearance: Minimum clearance > Hole to hole clearance(Different nets)
10. Minimum uVia diameter: Drill/Hole size > Min. Via hole size/diameter
11. Minimum uVia hole: Drill/Hole size > Min. Via hole size/diameter
12. Minimum item clearance: Legend > Pad To Silkscreen
13. Minimum text height: Legend > Minimum text height
14. Minimum text thickness: Legend > Minimum Line Width

Inspecting the Design for Errors

5. Open the Design Rules Checker window

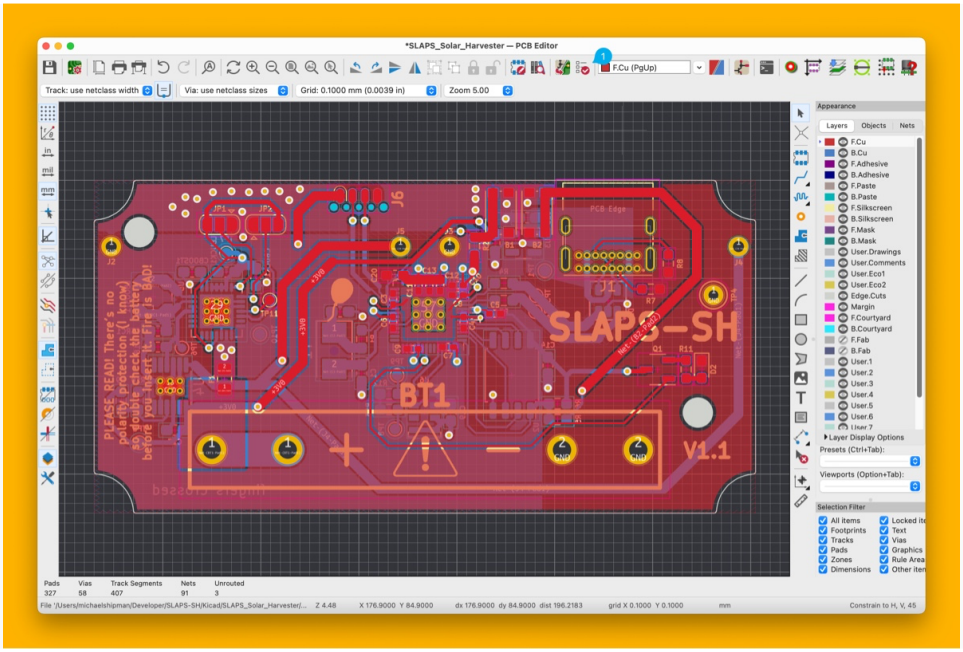

Figure 4

6. Select the options as seen in the image below and then run the checker to check for constraints violations in your design.

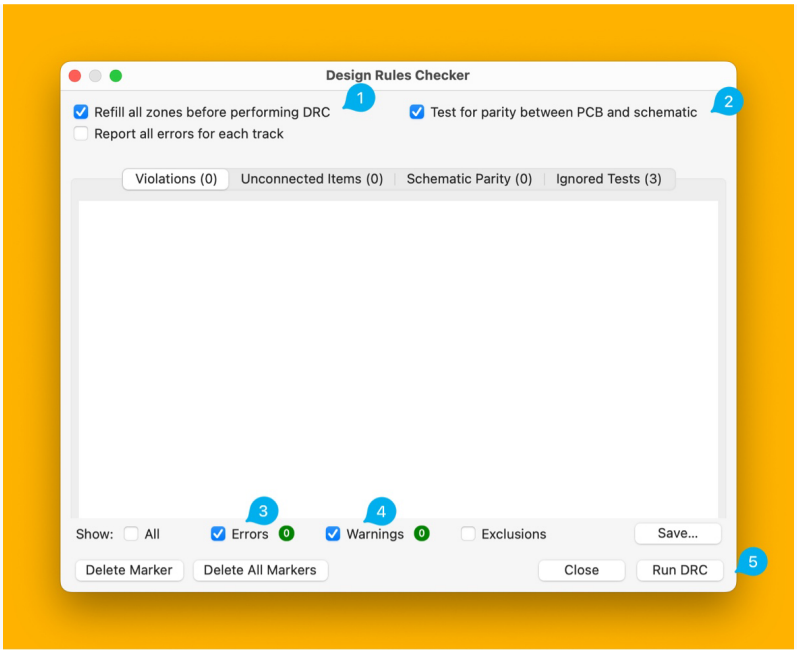

Figure 5

7. Each violation, warning, or other issue will appear in the DRC window as shown below. Clicking on an error or warning will highlight the issue in the PCB editor allowing you to find and fix the issue.

Note that violation definitions are not readily documented, in many cases the title of the violation will indicate the issue but further searching on the internet may be required. A good resource for searching and asking about violations is forum.kicad.info.

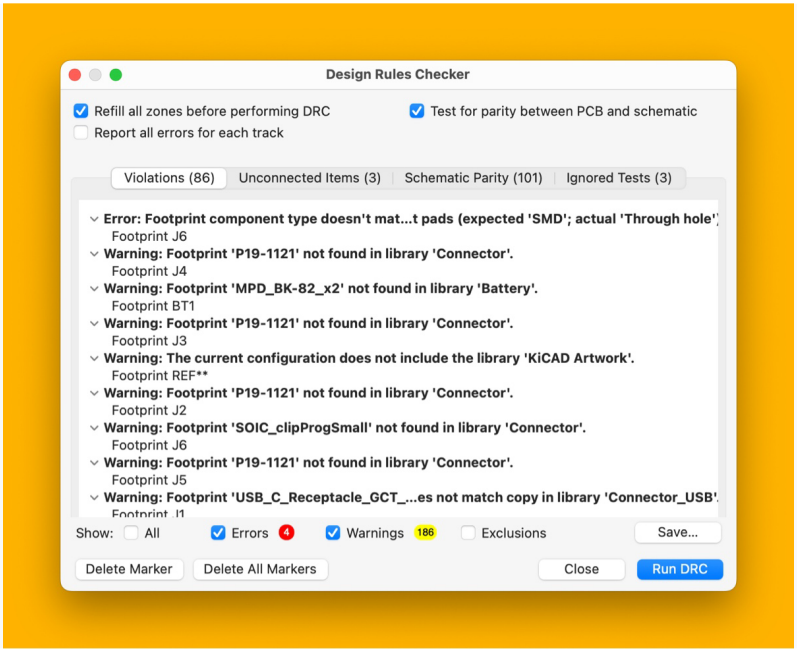

Figure 6

Violations - a type of error or warning that indicates a manufacturing constraint was violated

Unconnected Items - indicates an electrical connection that was specified in the schematic but has not been made in the board design

Schematic Parity - indicates either an electrical component in the schematic that is not included or an electrical component included on the PCB that is not in the schematic

Errors - these issues are likely to result in a board that cannot be made or one that will not function correctly

Warnings - these issues might result in a board that cannot be made or one that will not function correctly but are less severe than errors

Generating Gerber Files

8. Start by opening the Fabrication Outputs > Gerbers window

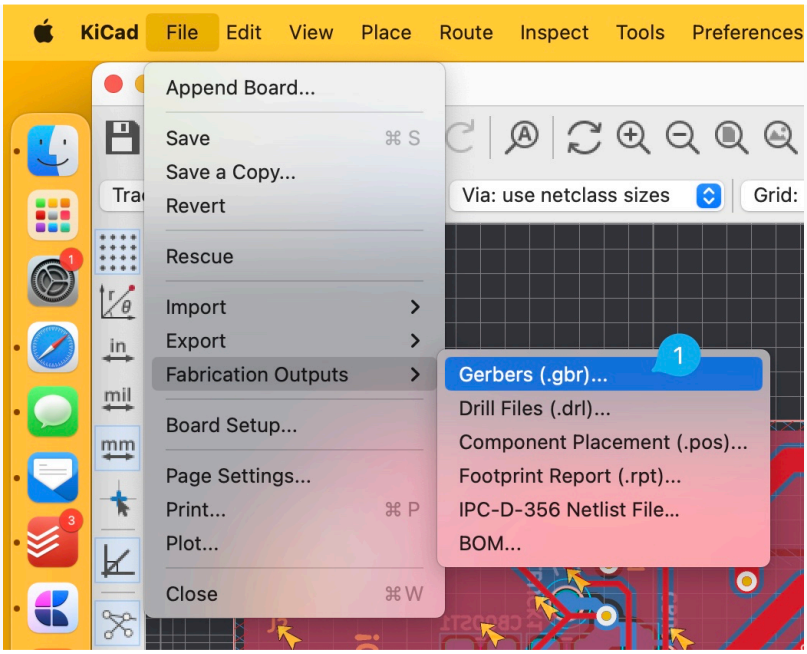

Figure 7

9. Select the following options as shown below.

You may use whatever folder you'd like at the `Output directory`, but it is recommended that you create a `Gerber` folder within the project folder to store the outputs as it makes the final product easier to order.

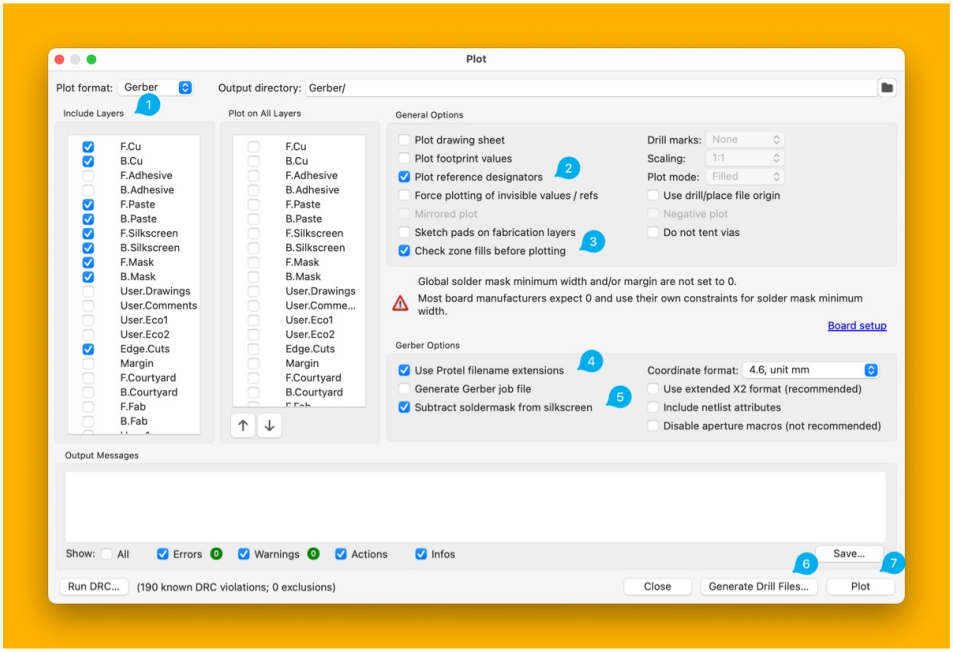

Figure 8

Generating Drill Files

10. Select the `Generate Drill Files` option, then select the options as shown below to generate the Drill and Maps files.

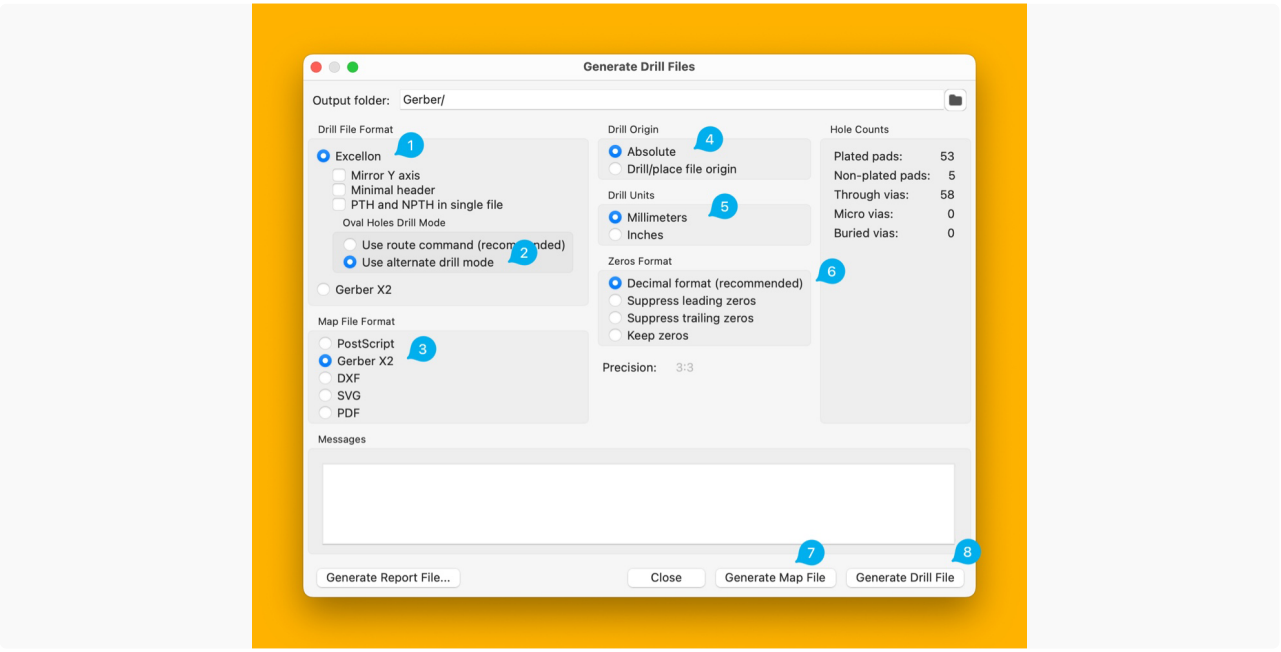

Figure 9

Ordering Your Design

Once all of your fabrication outputs have been created you are ready to create the final `.zip` for use at JLCPCB.

11. Navigate to the folder that you used to store your fabrication output files and compress the folder into a `.zip` file

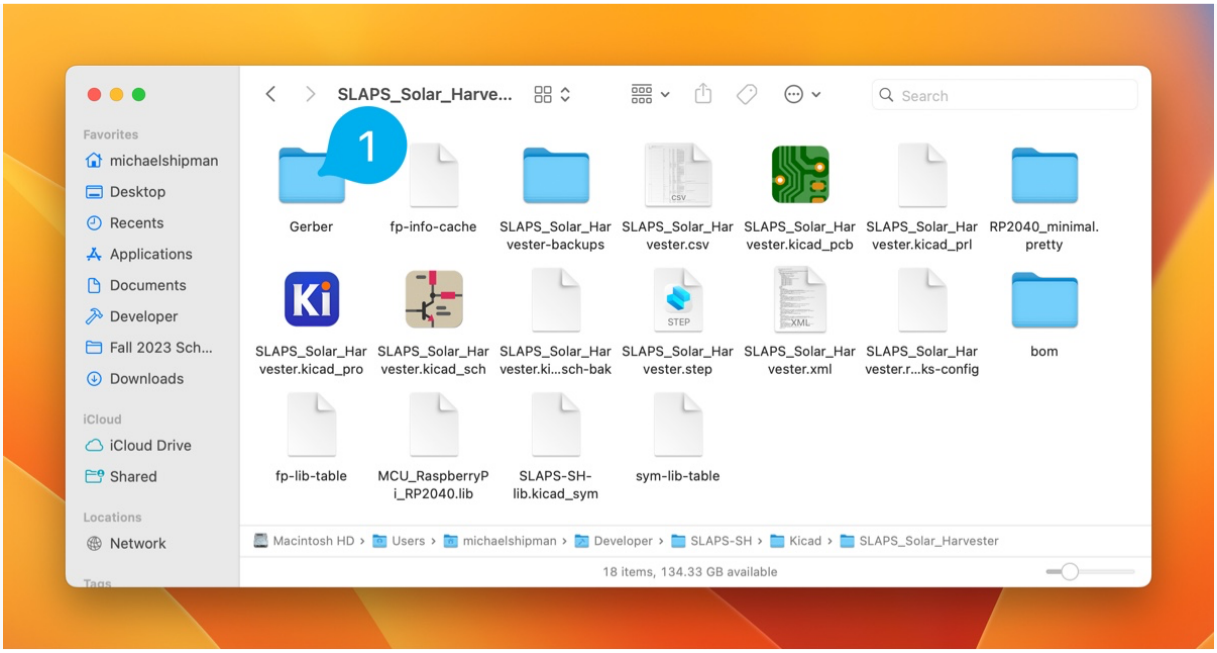

Figure 10

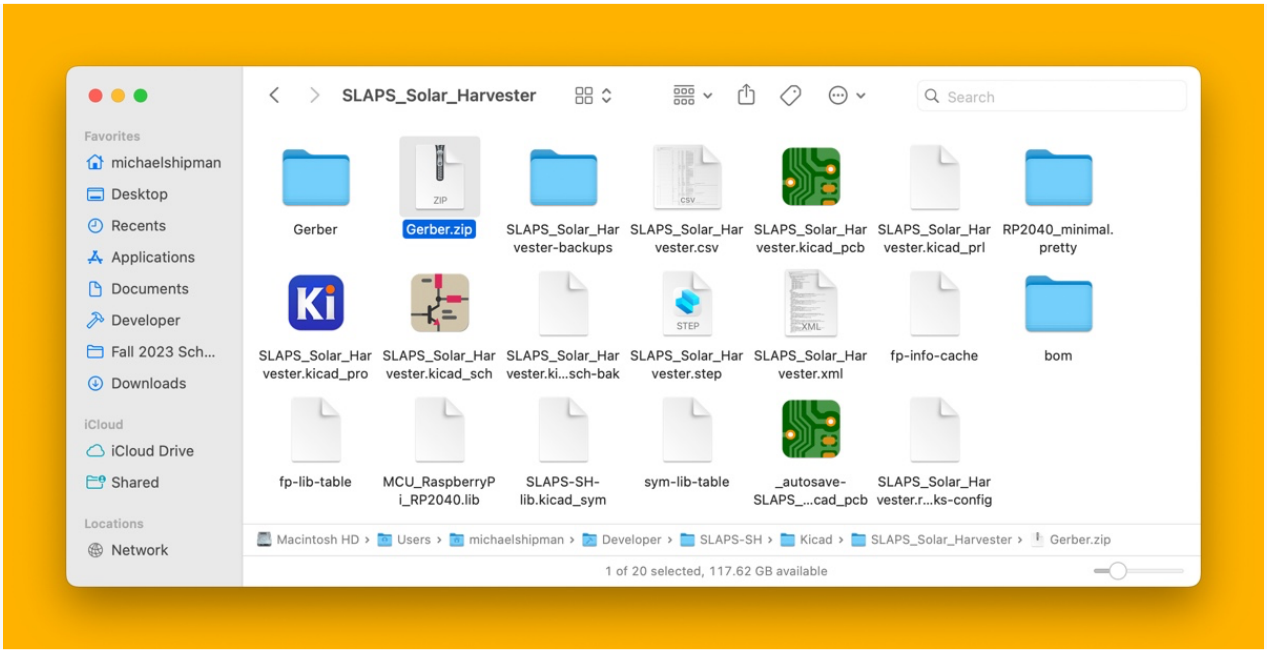

Figure 11

12. Upload your compressed Gerber to jlcpcb.com.

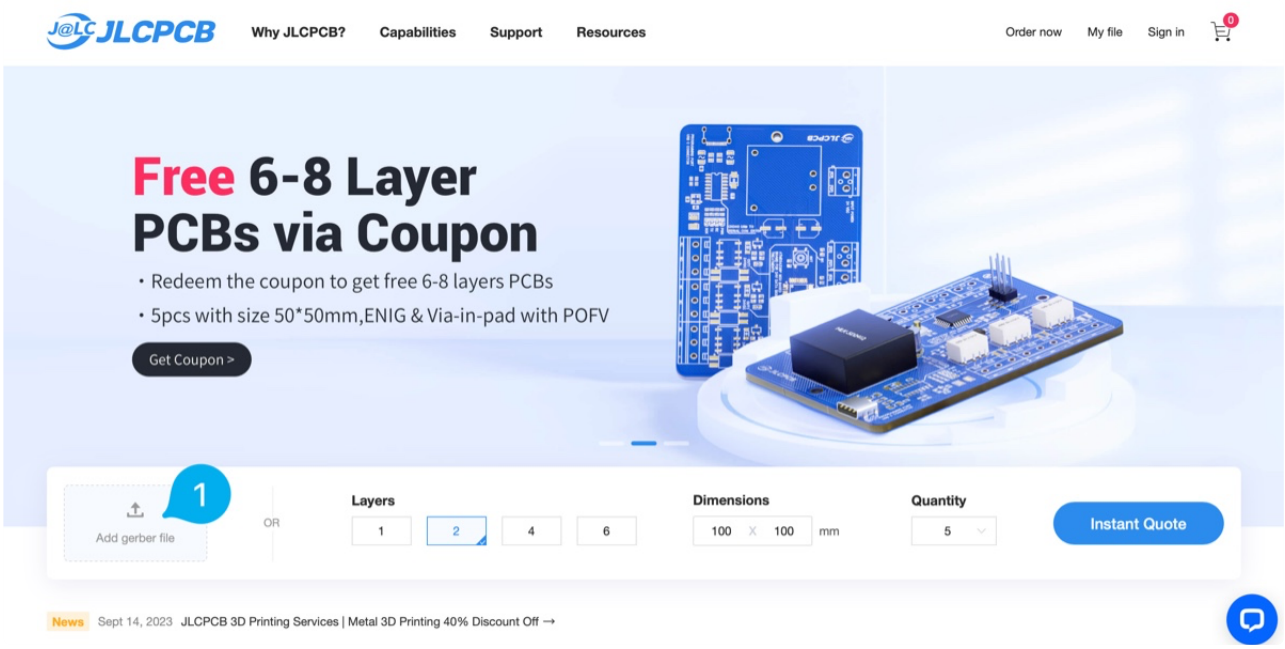

Figure 12

From here the default options selected will be adequate for most boards and you are free to order your design!

Note that each option that can be selected when ordering your design has a grey question mark next to that you can hover over for an explanation of what the option means.

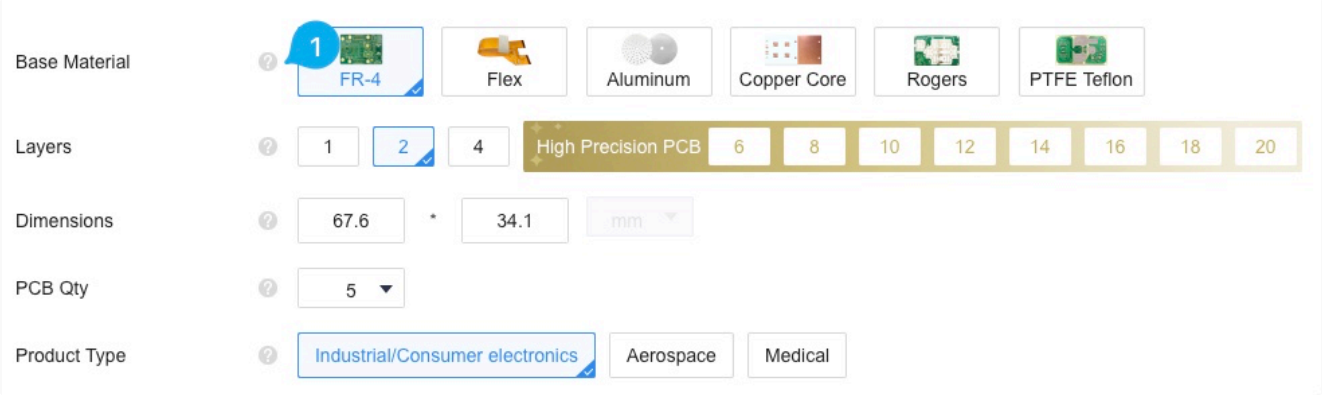

Figure 13

Bibliography

PCB Manufacturing & Assembly Capabilities. (n.d.). JLCPCB. Retrieved September 25, 2023, from https://jlcpcb.com/capabilities/pcb-capabilities

Design Rule Setup. (n.d.). OSHPARK. Retrieved September 25, 2023, from https://docs.oshpark.com/design-tools/kicad/kicad-design-rules/

Inspecting a board. (n.d.). KiCad Docs. Retrieved September 25, 2023, from https://docs.kicad.org/6.0/it/pcbnew/pcbnew_inspecting.html

How to generate Gerber and Drill files in KiCad 7. (2023, August 18). JLCPCB. Retrieved October 2, 2023, from https://jlcpcb.com/help/article/362-how-to-generate-gerber-and-drill-files-in-kicad-7

PCBShopper. (n.d.). PCBShopper. Retrieved October 2, 2023, from https://pcbshopper.com

JLCPCB. (n.d.). X. Retrieved October 2, 2023, from https://twitter.com/JLCPCB

lcamtuf. (2023, March 23). *Comparing hobby PCB vendors*. lcamtuf's thing. https://lcamtuf.substack.com/p/comparing-hobby-pcb-vendors

Stäbler, R. (2021, August 27). *PCB Design — A Hobbyist's Perspective*. Medium. https://raphaelstaebler.medium.com/pcb-design-a-hobbyists-perspective-68a3b713c8bc

Peterson, Z. (2020, October 5). What is a PCB and Intro to PCB Design. Altium. https://resources.altium.com/p/what-is-a-pcb.