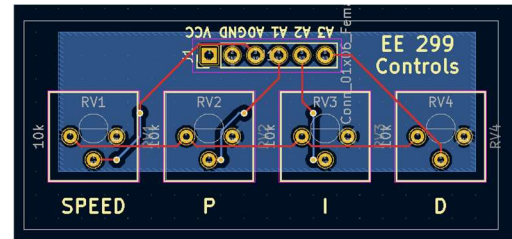
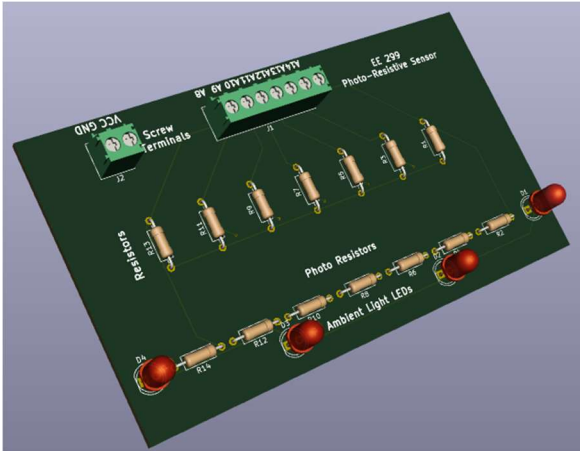


# Exporting and Ordering a PCB (KiCad)



Electrical and Computer Engineering

## General Guide to Exporting and Ordering a PCB

Exporting your design to Gerber Files and uploading to JLCPCB

---

# What You Will Need

---

## Materials:

- None

## Machinery:

- Computer / Laptop

## Software:

- KiCad Software Suite (Installed in Tutorial)

# Overview

The goal of this tutorial is to show you a quick (approx. 5 minute) way to export your designs from specified EDA suite to a format ready to be fabricated. We will be using [KiCad](#), but skills developed in this tutorial can be applied to analogous Electronic Design Automation (EDA) software suites such as [Eagle](#) and [Altium](#) (a few commonly used EDA suites in industry).

This tutorial is split into two portions: **(1) Exporting from KiCad** and **(2) Ordering from JLCPCB**. If you already have Gerber files and only need to order, skip to the “Ordering from JLCPCB” section.

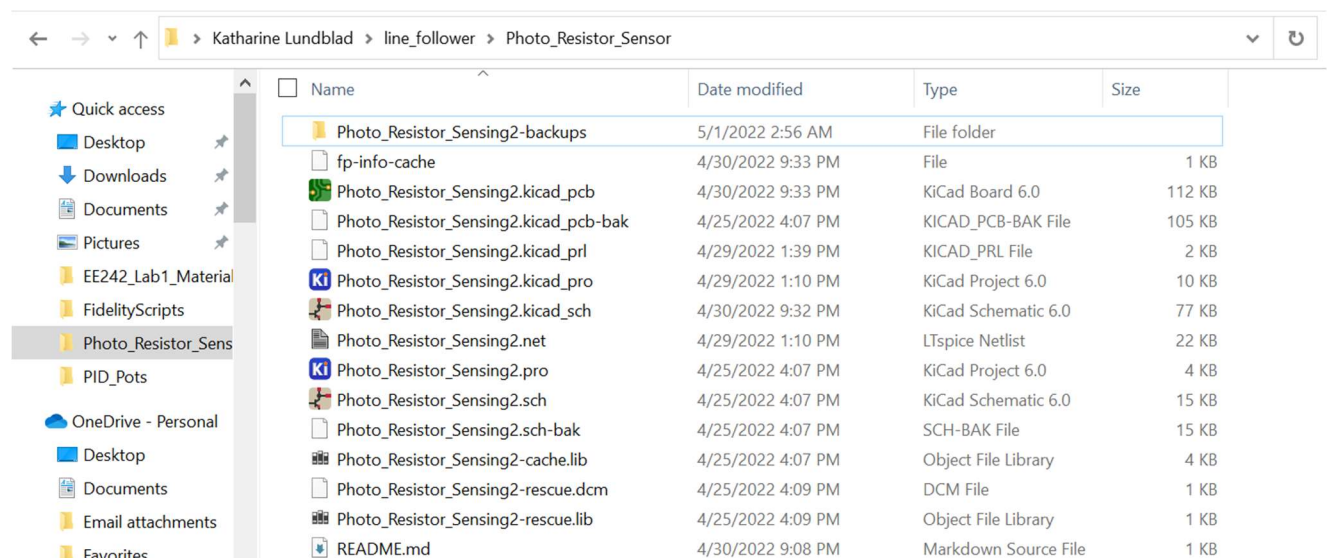
Note: KiCad continues to update their software, so this tutorial may resemble newer versions but may not completely match future versions. This version of the KiCad download guide is for the most recent **version 6.0.4**.

## Part 1: Exporting to Gerber Files

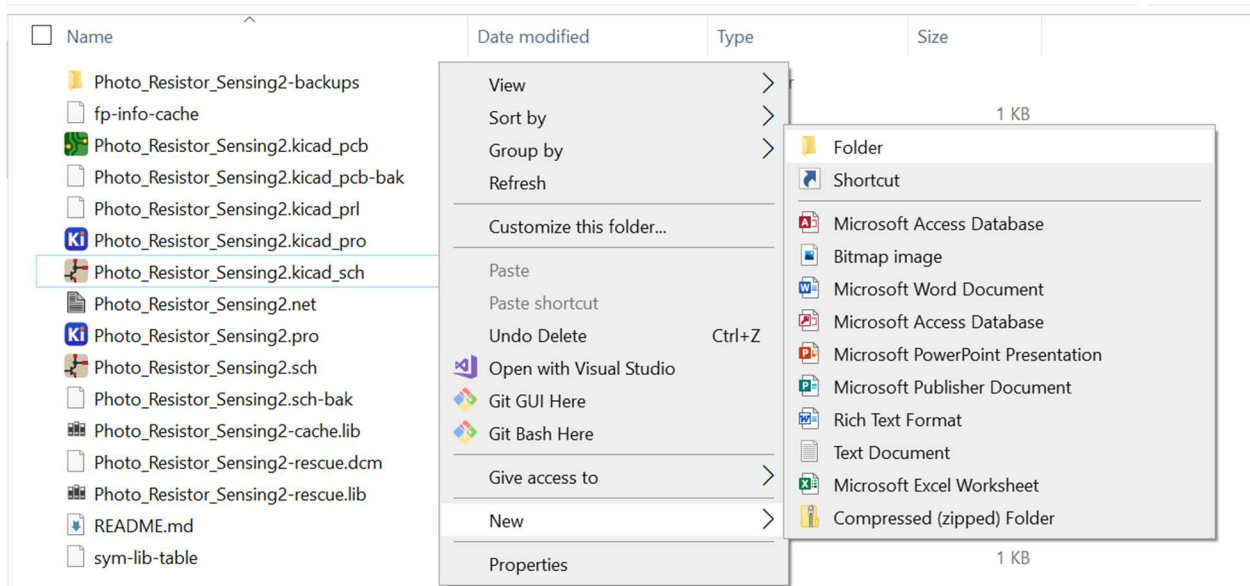
### Step 1 Create Output Directory:

Create a folder (ideally within your project folder) that will act as the output directory for the Gerber and Drill Files. Do this by

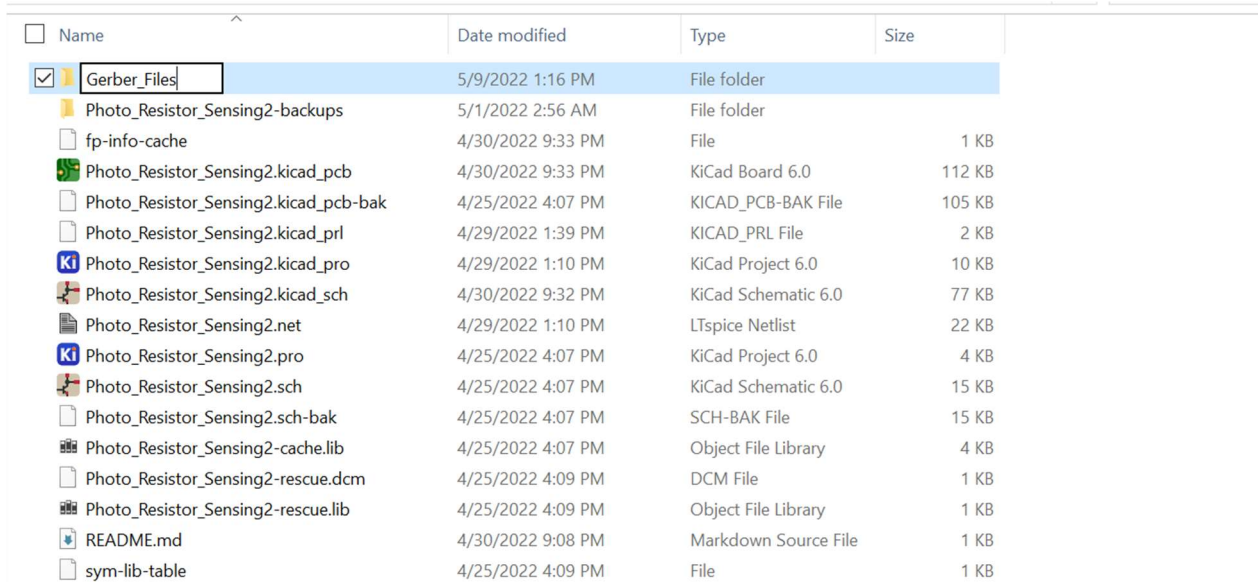
- (1) **Opening a location** within your project in your File Explorer,
- (2) **Right Clicking anywhere** in that directory
- (3) **Selecting New Folder**, and
- (4) **Naming it to something identifiable** like “Gerber\_Files”



**Figure 1: Open any directory you can easily access (I chose my KiCad Project Directory)**



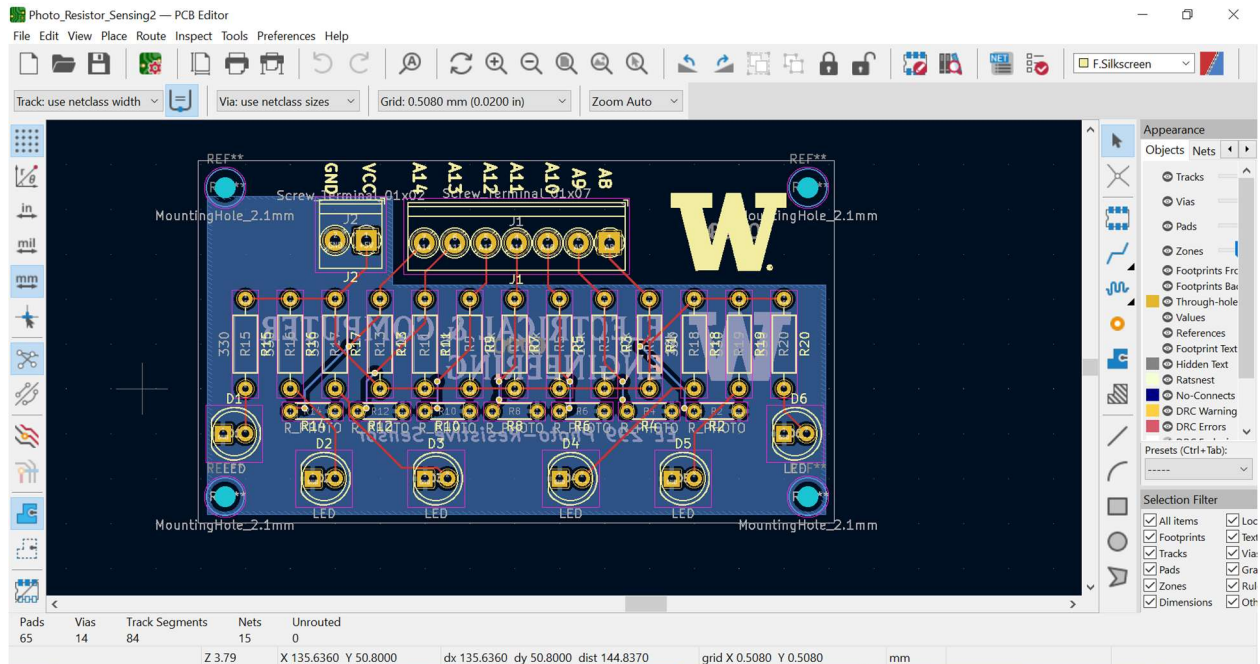
**Figure 2: Creating New Folder in Directory by Right Click > “New” > “Folder”**



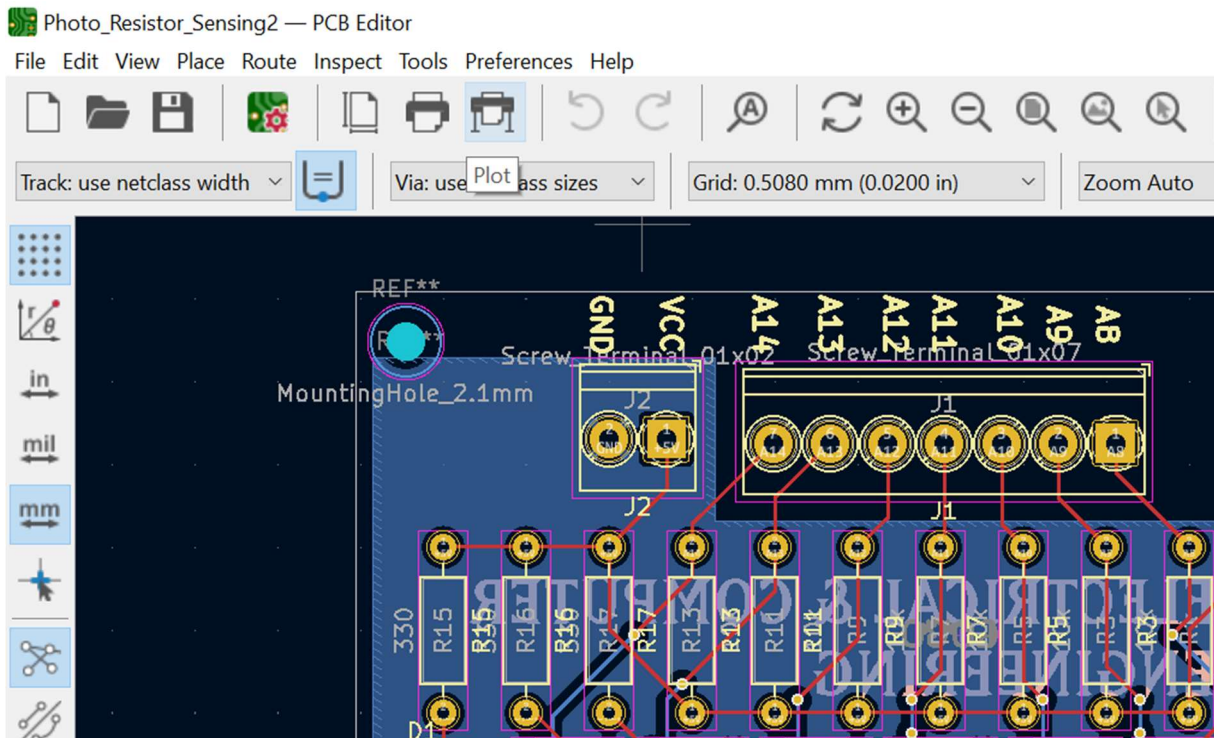
**Figure 3: Naming that folder to something like “Gerber\_Files”**

## Step 2 Plotting and Generating Drill Files:

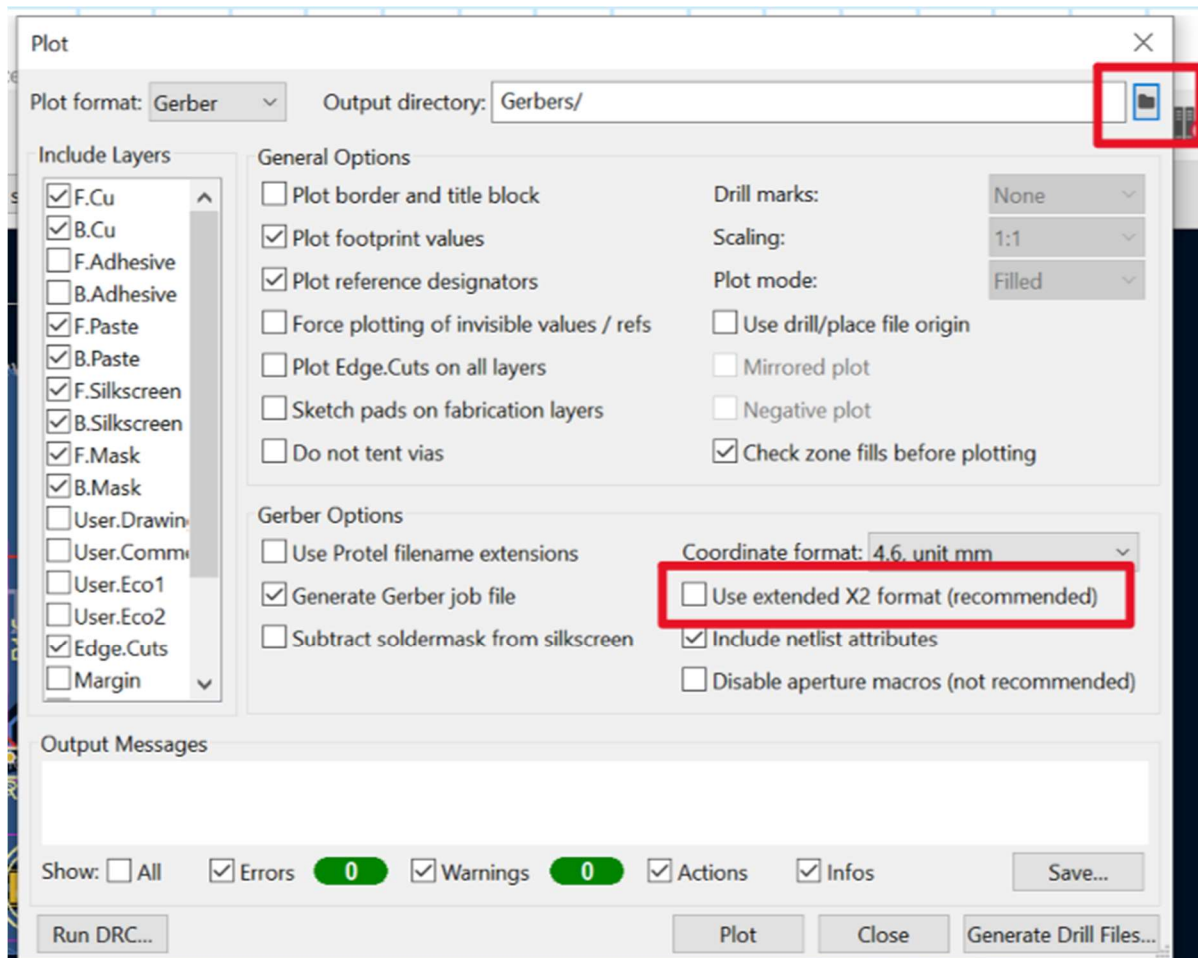
### (1) Open up your KiCad PCB Layout Editor:



### (2) Go to the Tools Bar and Select the “Plot” Icon (Looks like a Printer with Legs):

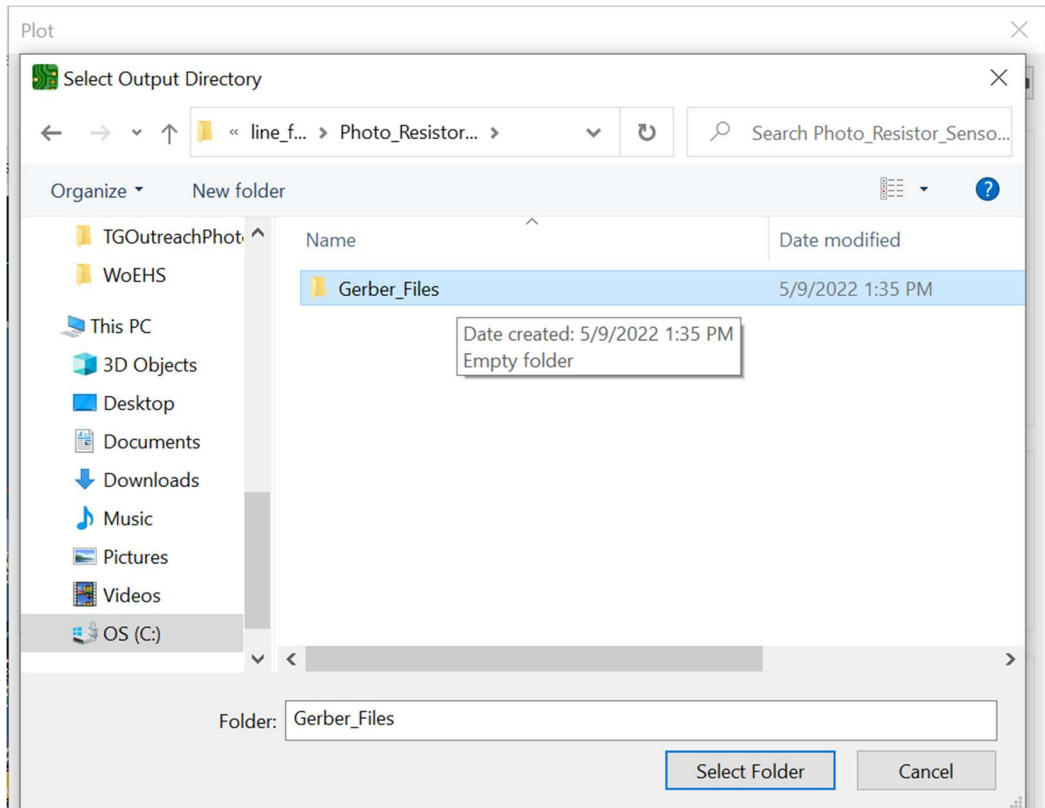


(3) This will Prompt the Plot window. Make sure you deselect “Use extended X2 format”, as JLCPCB does not support X2 formats. Then, select the folder icon next to “Output Directory”





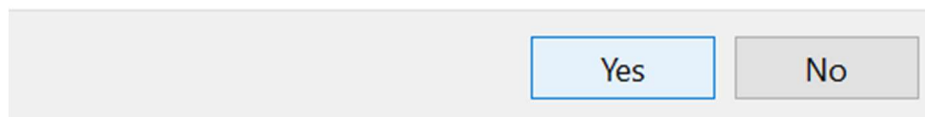
**(4) Select the folder you made earlier and then select “Select Folder”**



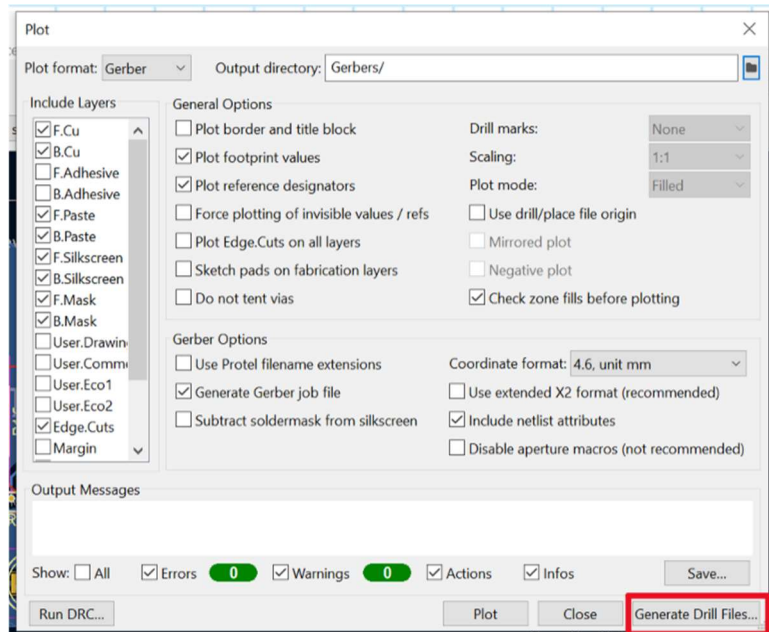
**(5) It will ask you confirm the path relative to your Plot Output Directory, Select “Yes”**

Plot Output Directory

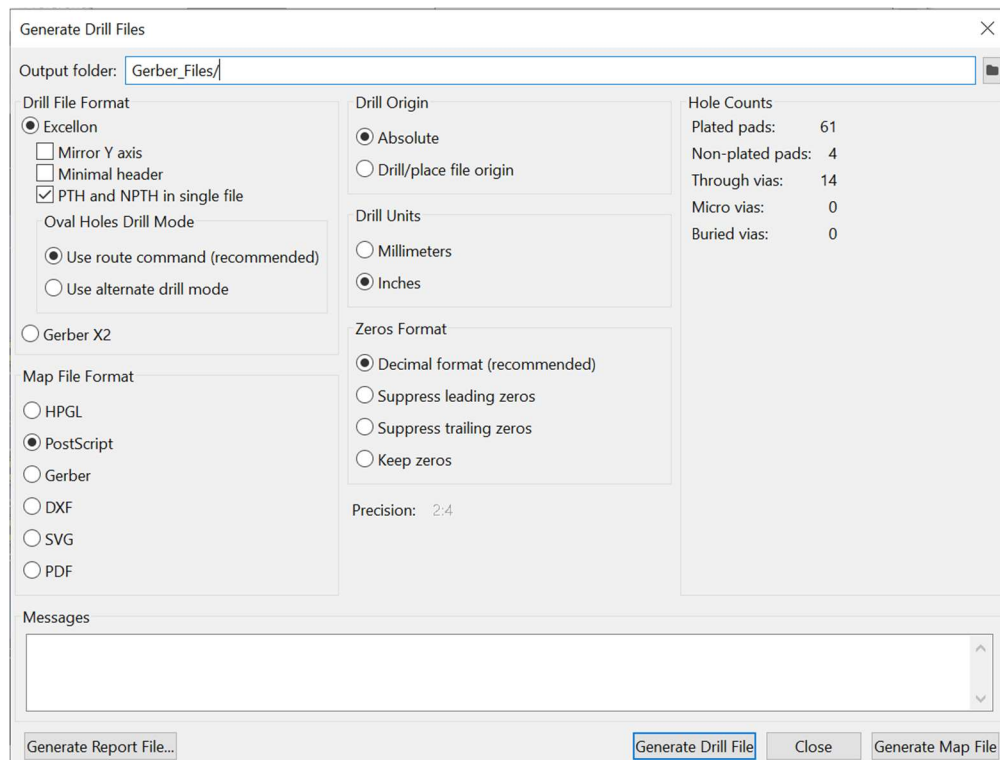
Do you want to use a path relative to  
'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\'?



**(6) Select “Generate Drill Files”. This will prompt a new window**



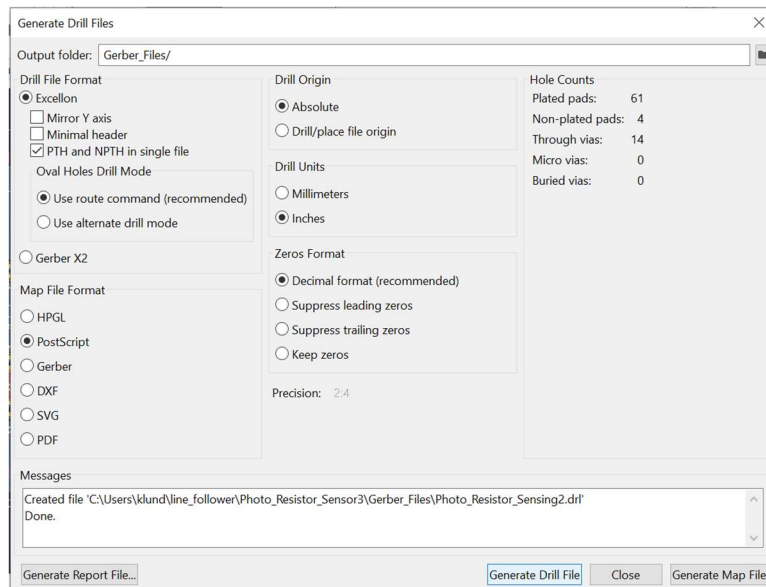
**(7) Make sure the Output Folder for the Drill Files is also the folder you created. Additionally, select “PTH and NPTH in single file”, and then click “Generate Drill File”**



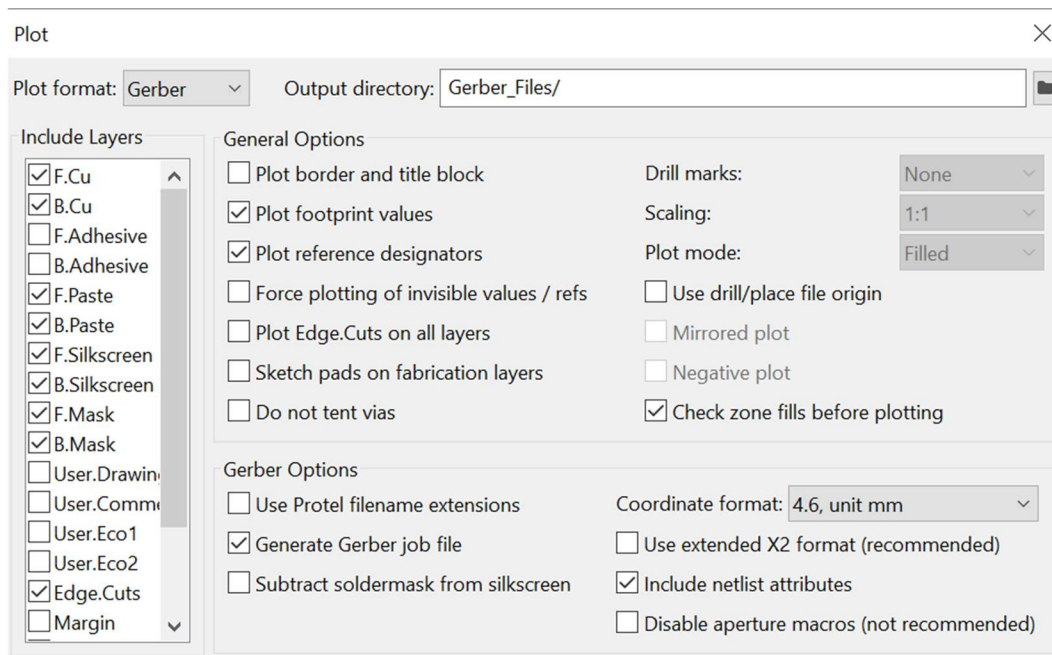
**(8) You should now see a new message in the “Messages” window indicating that a .drl file was created in the specified directory. It should also say “Done”.**



Then, select “Close”. You should be able to see this new .drl file in the new folder



(9) Make sure all of the layers in your PCB are included. They are commonly: “F.Cu”, “B.Cu”, “F.Paste”, “B.Paste”, “F.Silkscreen”, “B.Silkscreen”, “F.Mask”, “B.Mask”, and “Edge.Cuts”.



(10) Click “Plot”, and you should see one .gbr file for each layer of the PCB as well as a .gbrjob file. “Done” indicates all needed folders have been plotted

Plot

Plot format: Gerber

Output directory: Gerber\_Files/

Include Layers

☒ F.Cu

☒ B.Cu

☐ F.Adhesive

☐ B.Adhesive

☒ F.Paste

☒ B.Paste

☒ F.Silkscreen

☒ B.Silkscreen

☒ F.Mask

☒ B.Mask

☐ User.Drawings

☐ User.Comments

☐ User.Eco1

☐ User.Eco2

☒ Edge.Cuts

☐ Margin

General Options

☐ Plot border and title block

☒ Plot footprint values

☒ Plot reference designators

☐ Force plotting of invisible values / refs

☐ Plot Edge.Cuts on all layers

☐ Sketch pads on fabrication layers

☐ Do not tent vias

Drill marks: None

Scaling: 1:1

Plot mode: Filled

☐ Use drill/place file origin

☐ Mirrored plot

☐ Negative plot

☒ Check zone fills before plotting

Gerber Options

☐ Use Protel filename extensions

☒ Generate Gerber job file

☐ Subtract soldermask from silkscreen

Coordinate format: 4,6, unit mm

☐ Use extended X2 format (recommended)

☒ Include netlist attributes

☐ Disable aperture macros (not recommended)

Output Messages

Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-F\_Cu.gbr'.  
Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-B\_Cu.gbr'.  
Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-F\_Paste.gbr'.  
Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-B\_Paste.gbr'.  
Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-F\_Silkscreen.gbr'.  
Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-B\_Silkscreen.gbr'.  
Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-F\_Mask.gbr'.  
Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-B\_Mask.gbr'.  
Plotted to 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-Edge\_Cuts.gbr'.  
Created Gerber job file 'C:\Users\klund\line\_follower\Photo\_Resistor\_Sensor3\Gerber\_Files\Photo\_Resistor\_Sensing2-job.gbrjob'.  
Done.

Show: ☐ All ☒ Errors 0 ☒ Warnings 0 ☒ Actions ☒ Infos

Save...

Run DRC...

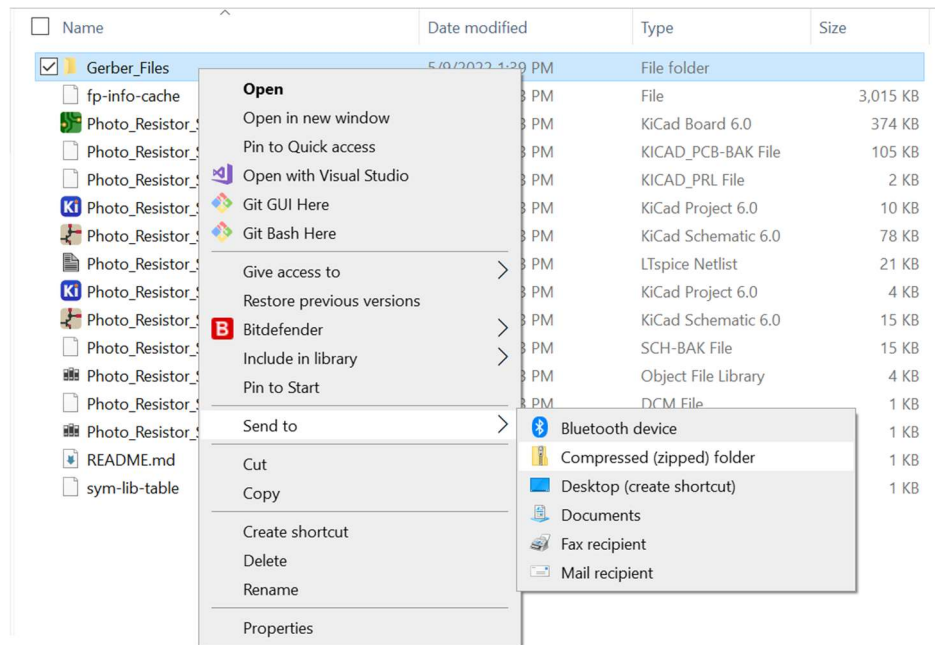
Plot

Close

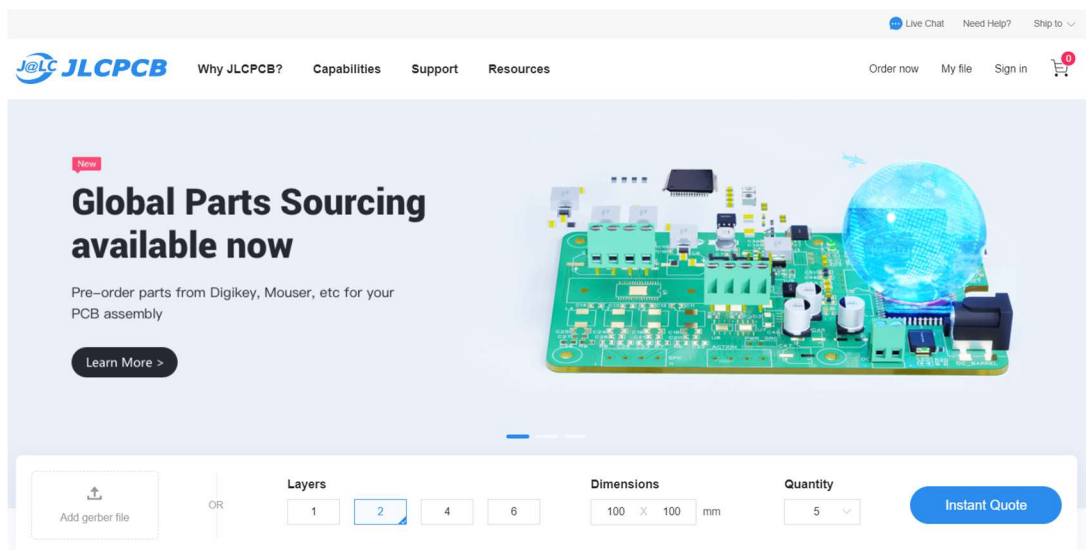
Generate Drill Files...

## Step 3 Compress into a .zip:

Go to the location of the folder containing the .gbr and .drl files and compress the folder into a .zip folder.



## Part 2: Ordering from JLCPCB




### Step 1 Website and Account Creation


(1) Go to [jlcpcb.com](http://jlcpcb.com) and select “Sign in”. Select “Sign up now”, and follow the steps to create an account

### Sign in to JLCPCB

Need new account? [Sign up](#) now

☐ I'm not a robot 

☐ Remember me [Forgot password?](#)

 [Sign in with Google](#)

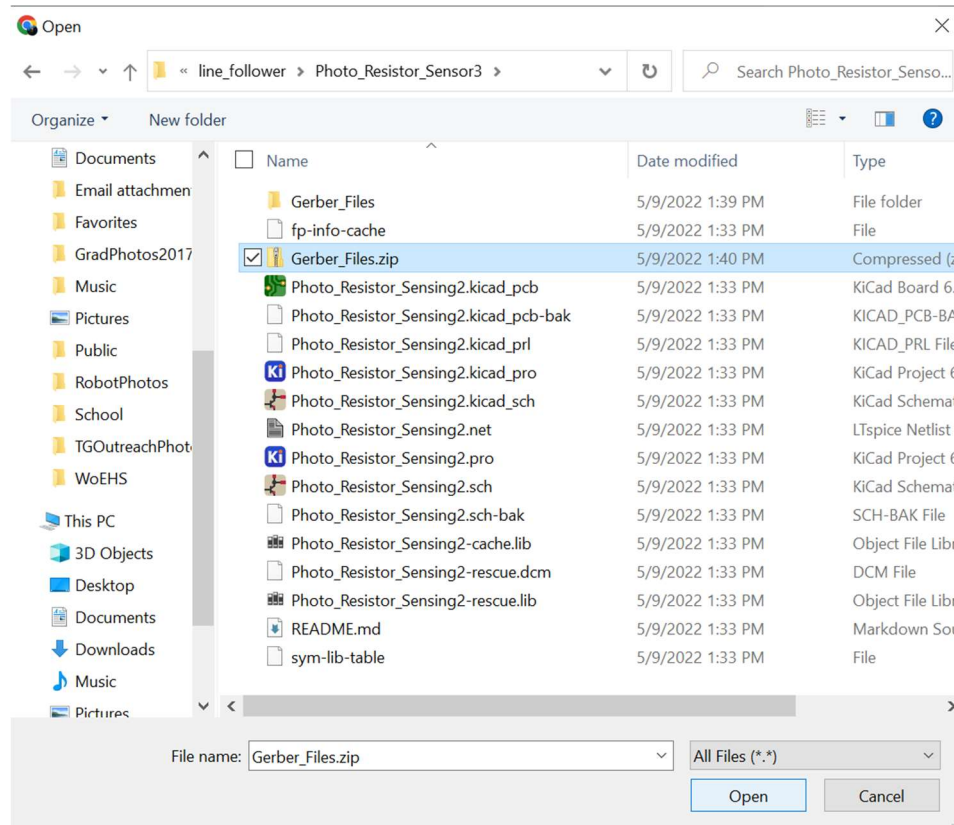
(2) Once you have logged into your new account, click “Order now” in the top right corner of the screen. You will be redirected to a page with a tab labeled “PCB”

The screenshot shows the JLCPCB website's PCB ordering interface. At the top, there is a navigation bar with the JLCPCB logo, links for 'Why JLCPCB?', 'Capabilities', 'Support', and 'Resources', and user options for 'Order now', 'My file', 'Sign in', and a shopping cart icon. A 'Live Chat' button is also present. Below the navigation bar, a large banner features the text 'Global Parts Sourcing available now' with a subtext 'Pre-order parts from Digikey, Mouser, etc for your PCB assembly' and a 'Learn More >' button. To the right of the text is an image of a green PCB with various components and a blue globe. Below the banner, there is a form with several input fields: 'Add gerber file' (with an upload icon), 'Layers' (with options 1, 2, 4, 6), 'Dimensions' (100 X 100 mm), and 'Quantity' (5). An 'Instant Quote' button is located to the right of the form.

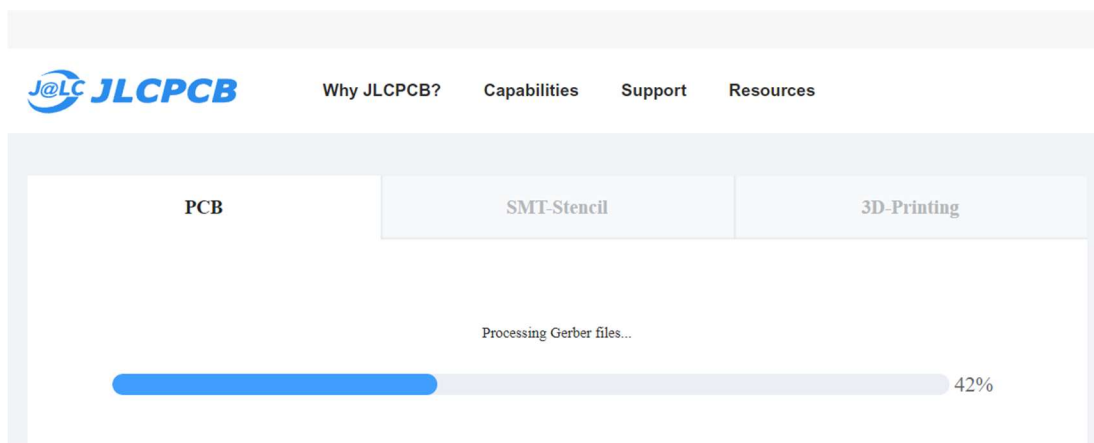
(3) Next, select “Add gerber file” which will prompt you to upload a .zip file

This screenshot shows the JLCPCB website's PCB ordering interface with the 'Add gerber file' button highlighted by a red rectangle. The interface includes a navigation bar at the top with the JLCPCB logo, links for 'Why JLCPCB?', 'Capabilities', 'Support', and 'Resources', and user options for 'Order now', 'My file', 'Sign in', and a shopping cart icon. A 'Live Chat' button is also present. Below the navigation bar, there are three tabs: 'PCB', 'SMT-Stencil', and '3D-Printing'. The 'PCB' tab is selected. In the center of the page, there is a large blue button labeled 'Add gerber file' with an upload icon. Below this button, there is a note: 'Only accept zip or rar, Max 10 M, View example >'. To the right of the button, there are links for 'Instructions for ordering' and 'Log in to view your upload history'. Below the button, there are input fields for 'Base Material' (FR-4, Aluminum), 'Layers' (1, 2, 4, 6), and 'Dimensions' (100 X 100 mm). On the right side of the page, there is a 'Charge Details' section with a 'Special Offer' of \$2.00, 'Build Time' of 1-2 days, and a 'Calculated Price' of \$4.00 - \$2.00. Below this, there is a 'Weight' of 0.28kg and a 'SAVE TO CART' button. At the bottom, there is a 'Shipping Estimate' section with a 'Charge' of 'Choose destination country first'.

#### (4) Select your compressed Gerber\_Files.zip folder and click “Open”

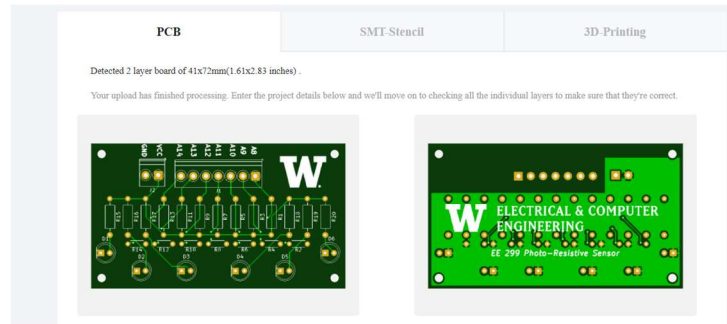


You will then see a prompt to load the Gerber files from your compressed folder



After uploading, JLCPCB should generate a render of your PCB design





**(5) The number of layers and the dimensions of your PCB will be autogenerated. It is ideal to have a Lead-Free board, so select “LeadFree HASL-RoHS”**

← Back to Upload File Gerber Viewer

Base Material: ☐ FR-4 ☐ Aluminum

Layers: ☐ 1 ☒ 2 ☐ 4 ☐ 6

Dimensions:  \*

PCB Qty:

Product Type: ☒ Industrial/Consumer electronics ☐ Military/Aerospace ☐ Medical

Different Design: ☒ 1 ☐ 2 ☐ 3 ☐ 4 ☐

Delivery Format: ☒ Single PCB ☐ Panel by Customer ☐ Panel by JLCPCB

PCB Thickness: ☐ 0.4 ☐ 0.6 ☐ 0.8 ☐ 1.0 ☐ 1.2 ☒ 1.6 ☐ 2.0

PCB Color: ☒ Green ☐ Purple ☐ Red ☐ Yellow ☐ Blue ☐ White ☐ Black

Silkscreen: ☒ White ☐

Surface Finish: ☐ HASL(with lead) ☒ LeadFree HASL-RoHS ☐ ENIG-RoHS

Outer Copper Weight: ☒ 1 oz ☐ 2 oz

Gold Fingers: ☒ No ☐ Yes

Confirm Production file: ☒ No ☐ Yes

Flying Probe Test: ☒ Fully Test ☐ Not Test

Castellated Holes: ☒ No ☐ Yes

Remove Order Number: ☒ No ☐ Yes

Advanced Options

**Feel free to also choose a color that you like for the PCB although choosing any color other than green will take two more days to fabricate than the initial time.**

<b>Charge Details</b> ^	
Special Offer	\$2.00
<hr/>	
Build Time ?	
PCB: ● 1-2 days	\$0.00
<hr/>	
<b>Calculated Price</b>	<del>\$4.00</del> <b>\$2.00</b>
Additional charges may apply for <a href="#">special cases</a>	
Weight ?	0.16kg
<div>SAVE TO CART</div>	
Shipping Estimate	
Charge: <a href="#">Choose destination country first</a>	

**(6) Finally, save the PCB to your cart, and follow the steps to checkout and ship your PCB to your location!**

**Congratulations!**  
**You have successfully exported and uploaded your Printed Circuit Board to a fabrication service!**