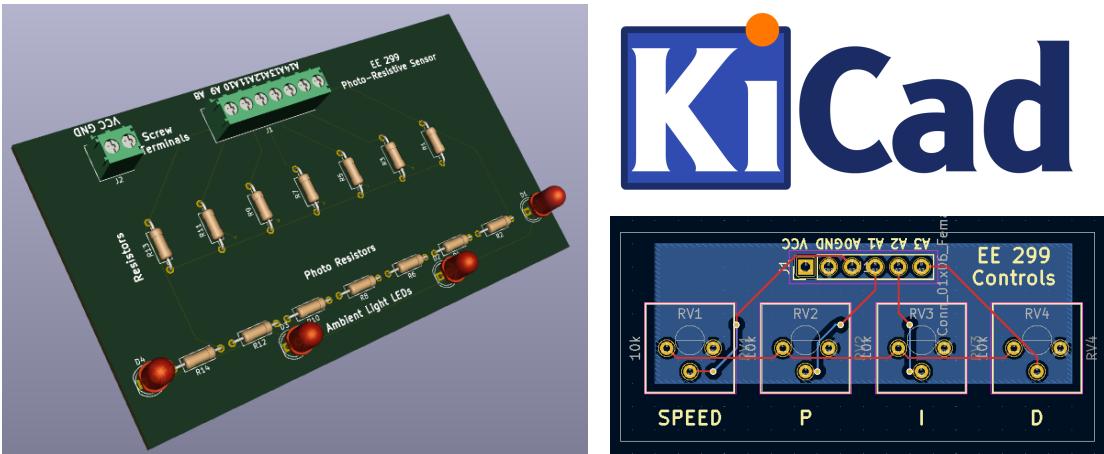


Exporting and Ordering a PCB (KiCad)



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 four 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”

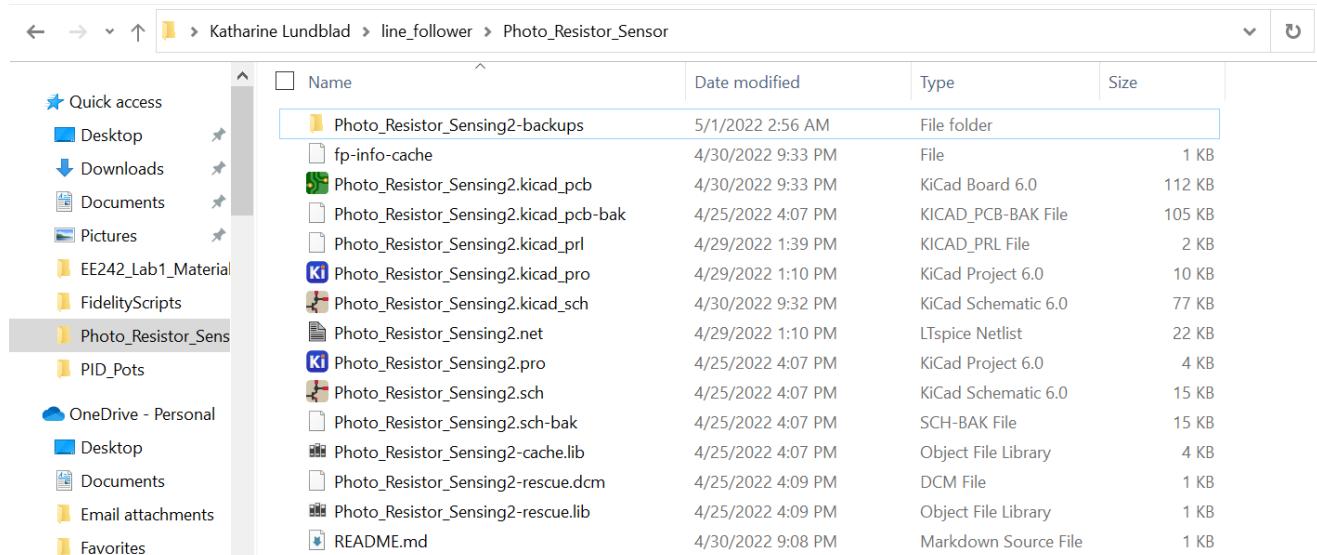


Figure 1: Open any directory you can easily access (I chose my KiCad Project Directory)

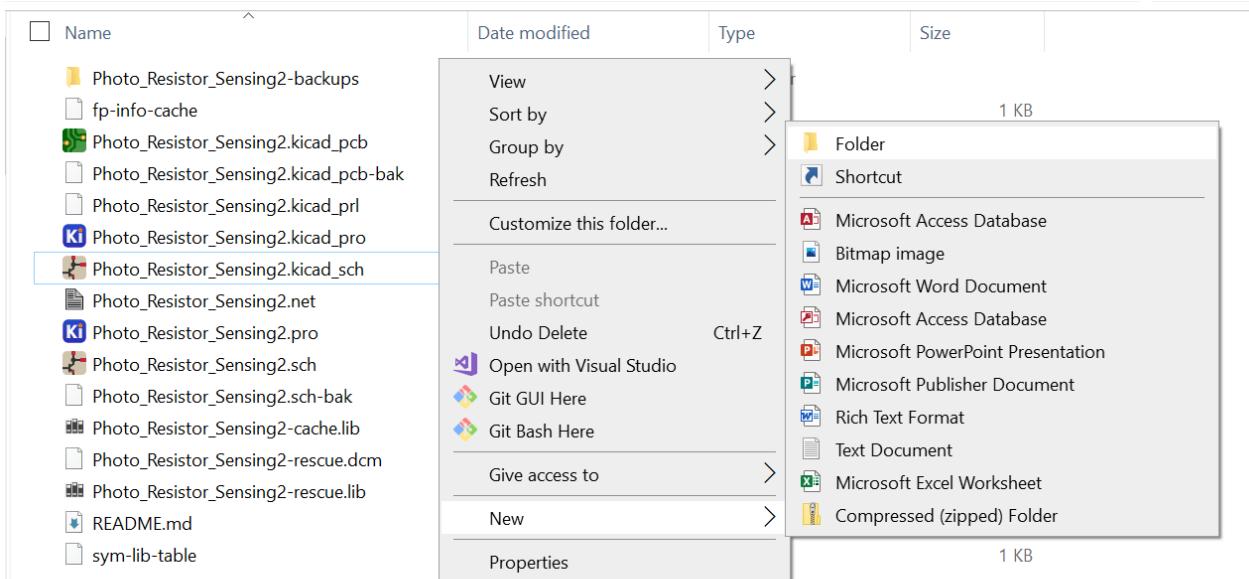


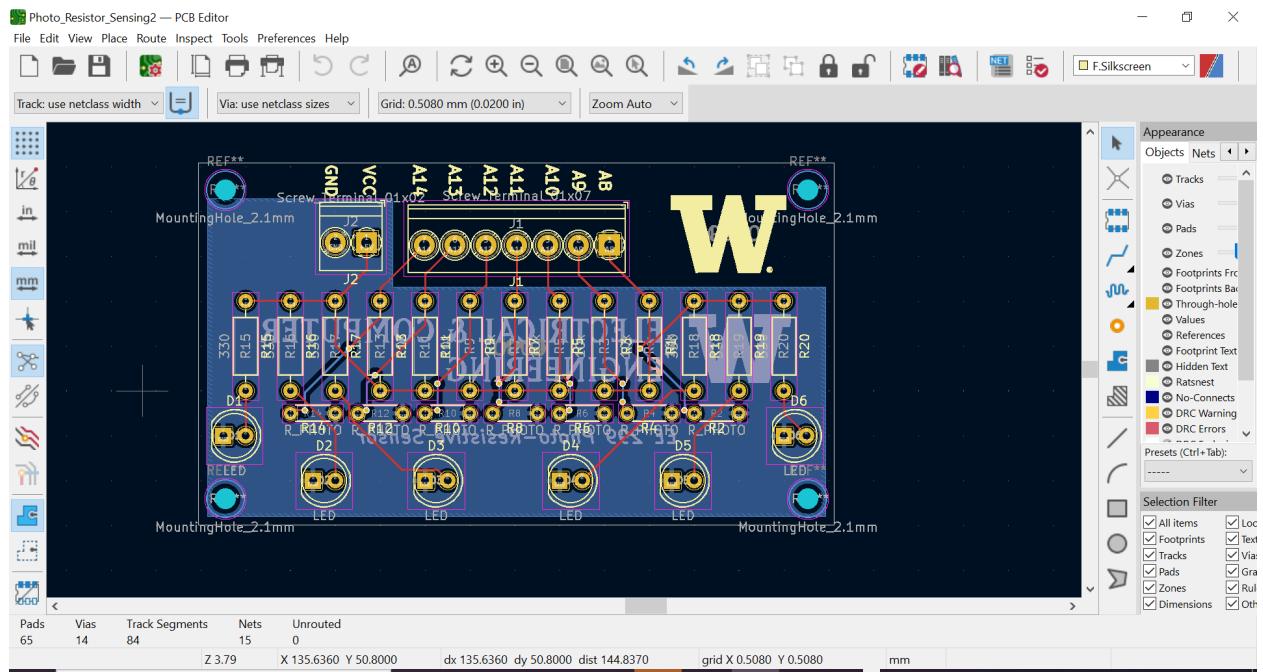
Figure 2: Creating New Folder in Directory by Right Click > “New” > “Folder”

Name	Date modified	Type	Size
<input checked="" type="checkbox"/> Gerber_Files	5/9/2022 1:16 PM	File folder	
Photo_Resistor_Sensing2-backups	5/1/2022 2:56 AM	File folder	
fp-info-cache	4/30/2022 9:33 PM	File	1 KB
Photo_Resistor_Sensing2.kicad_pcb	4/30/2022 9:33 PM	KiCad Board 6.0	112 KB
Photo_Resistor_Sensing2.kicad_pcb-bak	4/25/2022 4:07 PM	KICAD_PCB-BAK File	105 KB
Photo_Resistor_Sensing2.kicad_prl	4/29/2022 1:39 PM	KICAD_PRL File	2 KB
Photo_Resistor_Sensing2.kicad_pro	4/29/2022 1:10 PM	KiCad Project 6.0	10 KB
Photo_Resistor_Sensing2.kicad_sch	4/30/2022 9:32 PM	KiCad Schematic 6.0	77 KB
Photo_Resistor_Sensing2.net	4/29/2022 1:10 PM	LTspice Netlist	22 KB
Photo_Resistor_Sensing2.pro	4/25/2022 4:07 PM	KiCad Project 6.0	4 KB
Photo_Resistor_Sensing2.sch	4/25/2022 4:07 PM	KiCad Schematic 6.0	15 KB
Photo_Resistor_Sensing2.sch-bak	4/25/2022 4:07 PM	SCH-BAK File	15 KB
Photo_Resistor_Sensing2-cache.lib	4/25/2022 4:07 PM	Object File Library	4 KB
Photo_Resistor_Sensing2-rescue.dcm	4/25/2022 4:09 PM	DCM File	1 KB
Photo_Resistor_Sensing2-rescue.lib	4/25/2022 4:09 PM	Object File Library	1 KB
README.md	4/30/2022 9:08 PM	Markdown Source File	1 KB
sym-lib-table	4/25/2022 4:09 PM	File	1 KB

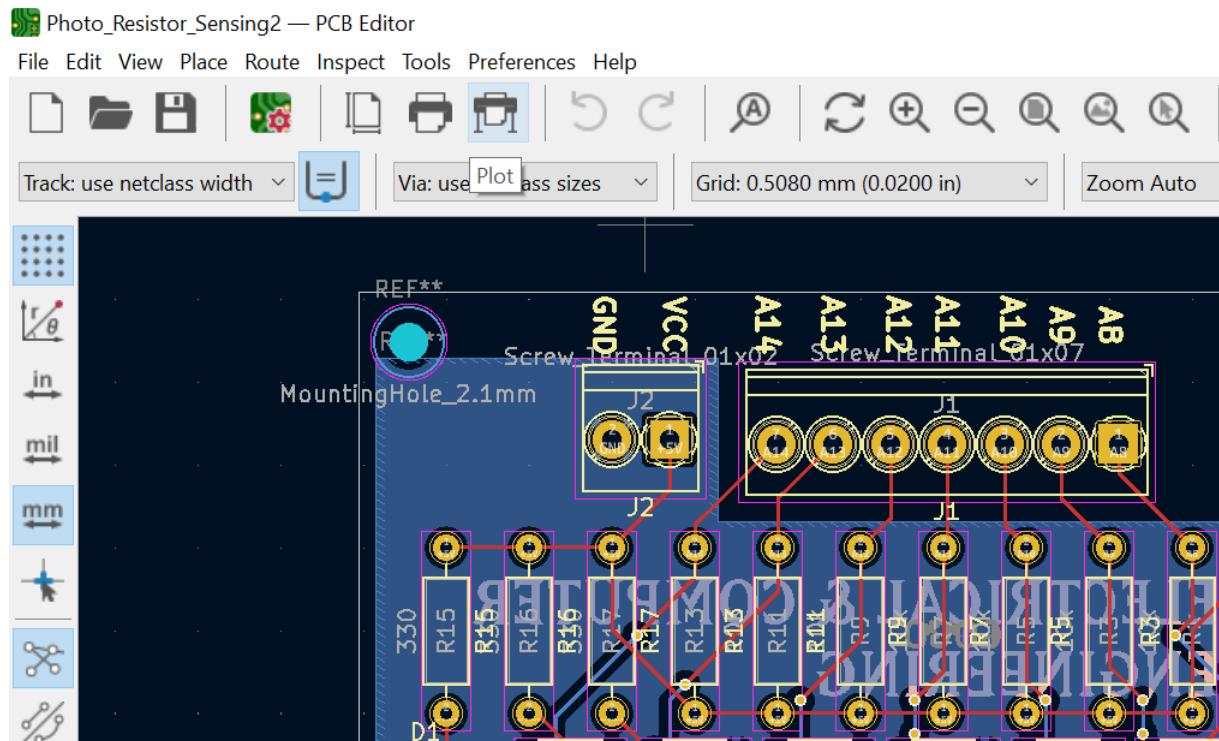
Figure 3: Naming that folder to something like “Gerber_Files”

Step 2 Plotting and Generating Drill Files:

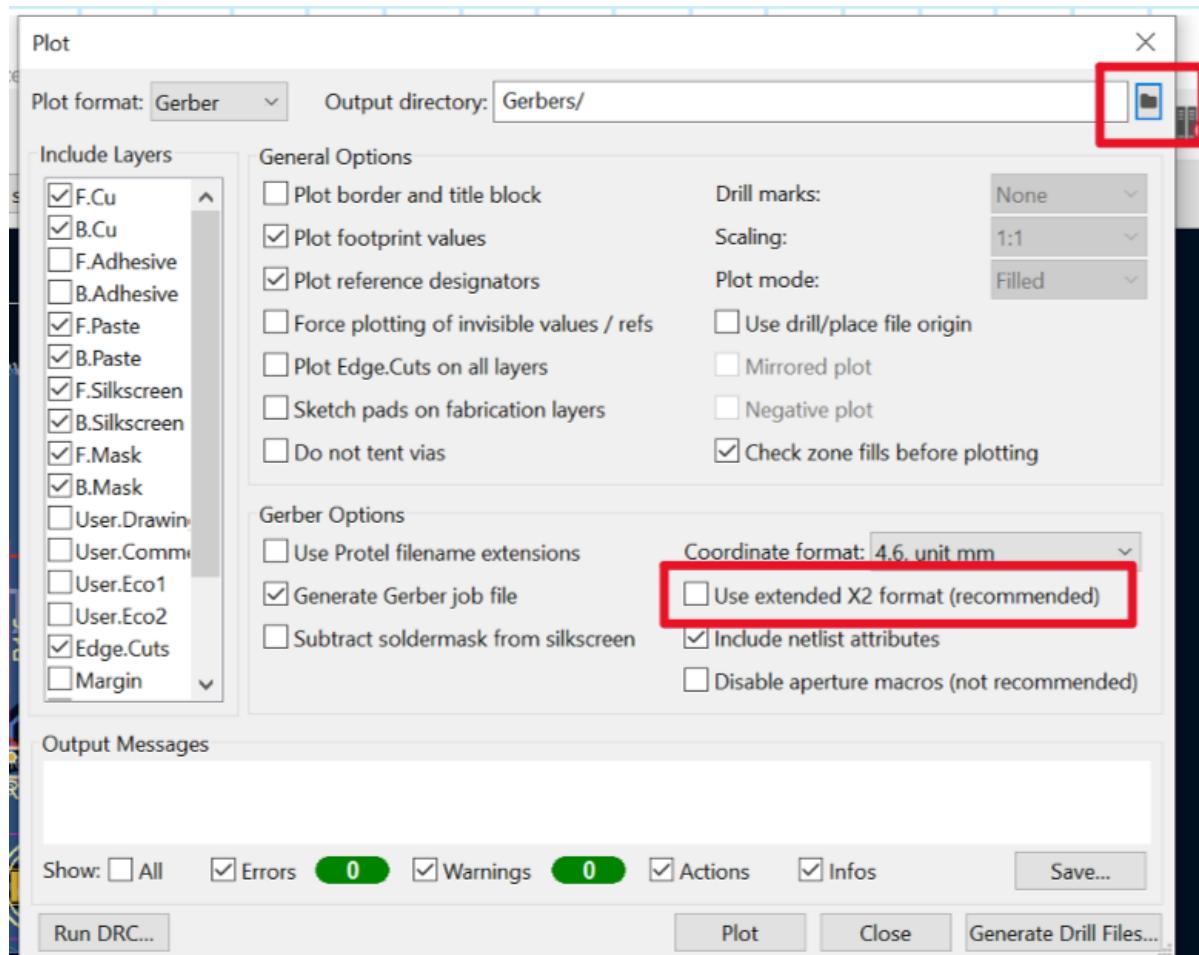
(1) Open up your KiCad PCB Layout Editor:



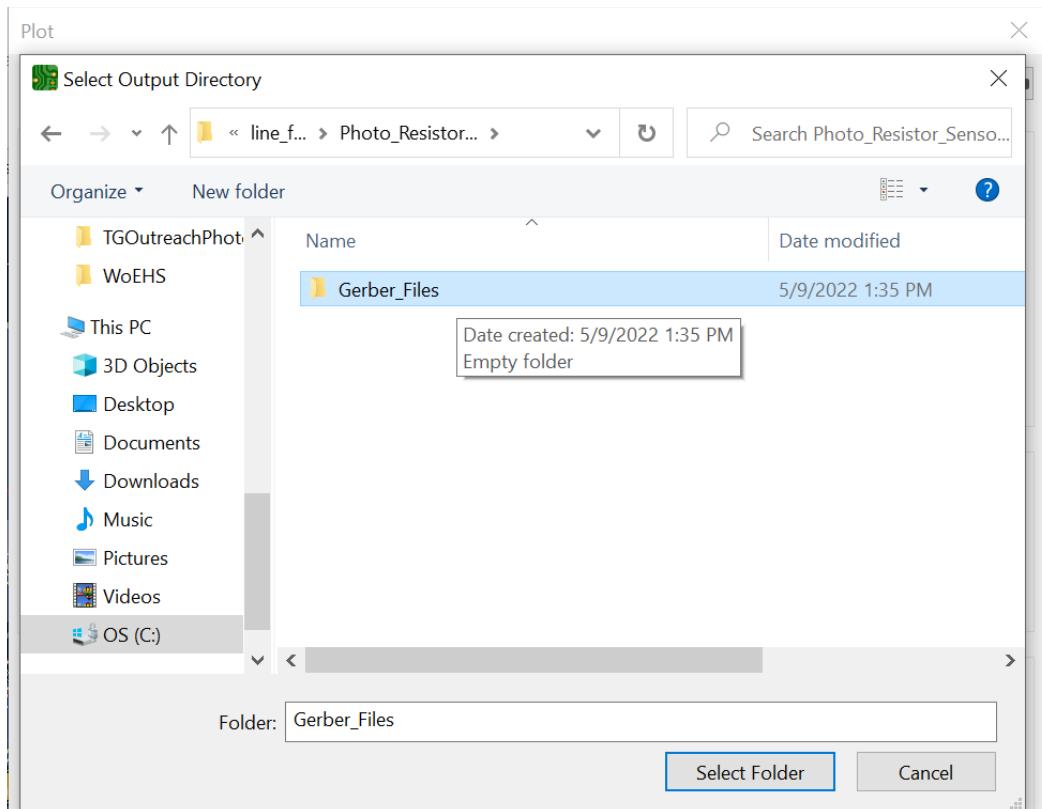
(2) Go to the Tools Bar and Select the “Plot” Icon (Looks like a Printer with Legs):



(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”



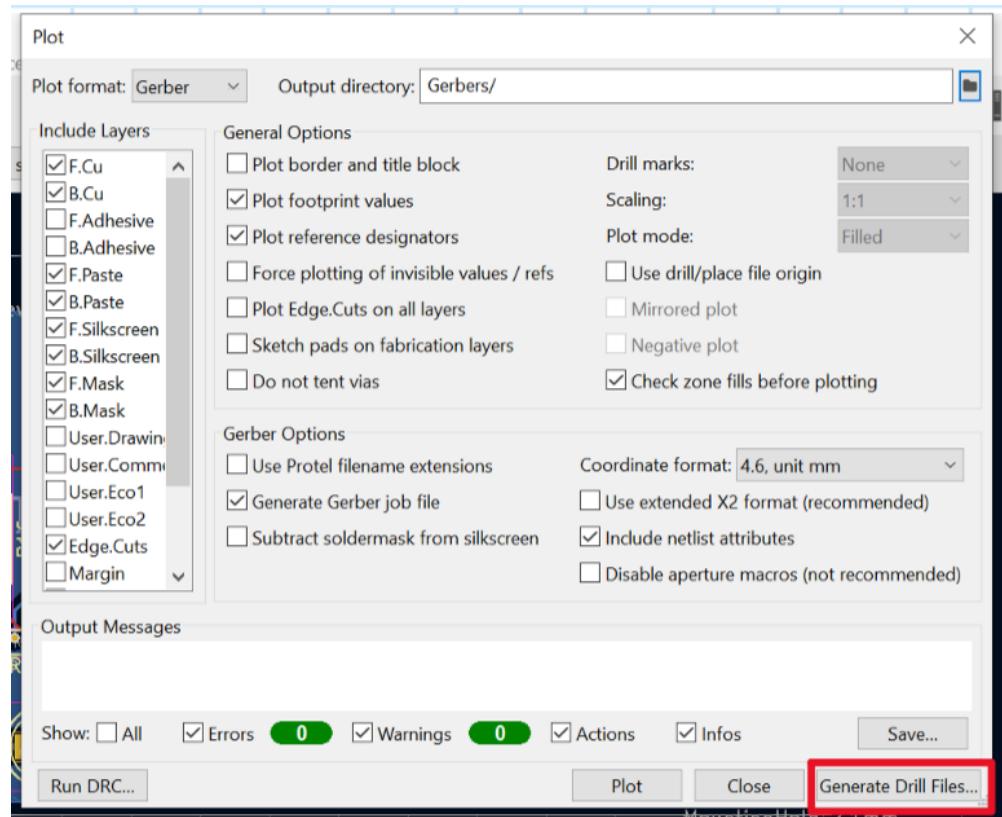
(5) It will ask you confirm the path relative to your Plot Output Directory, Select “Yes”

Plot Output Directory

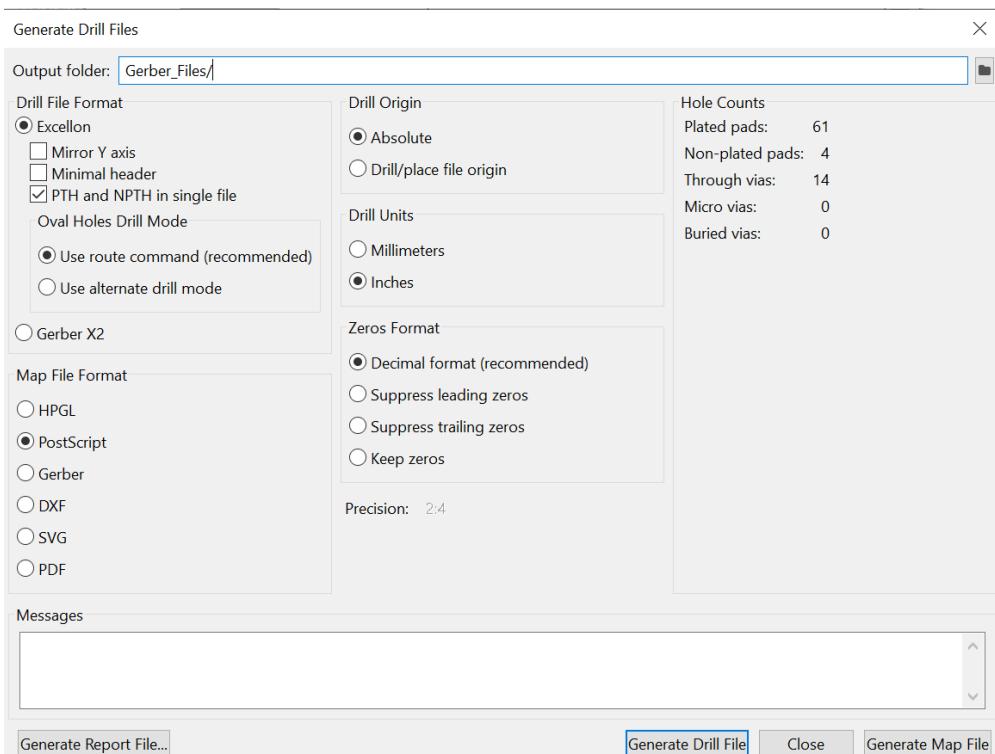
Do you want to use a path relative to
'C:\Users\klund\line_follower\Photo_Resistor_Sensor3\'?



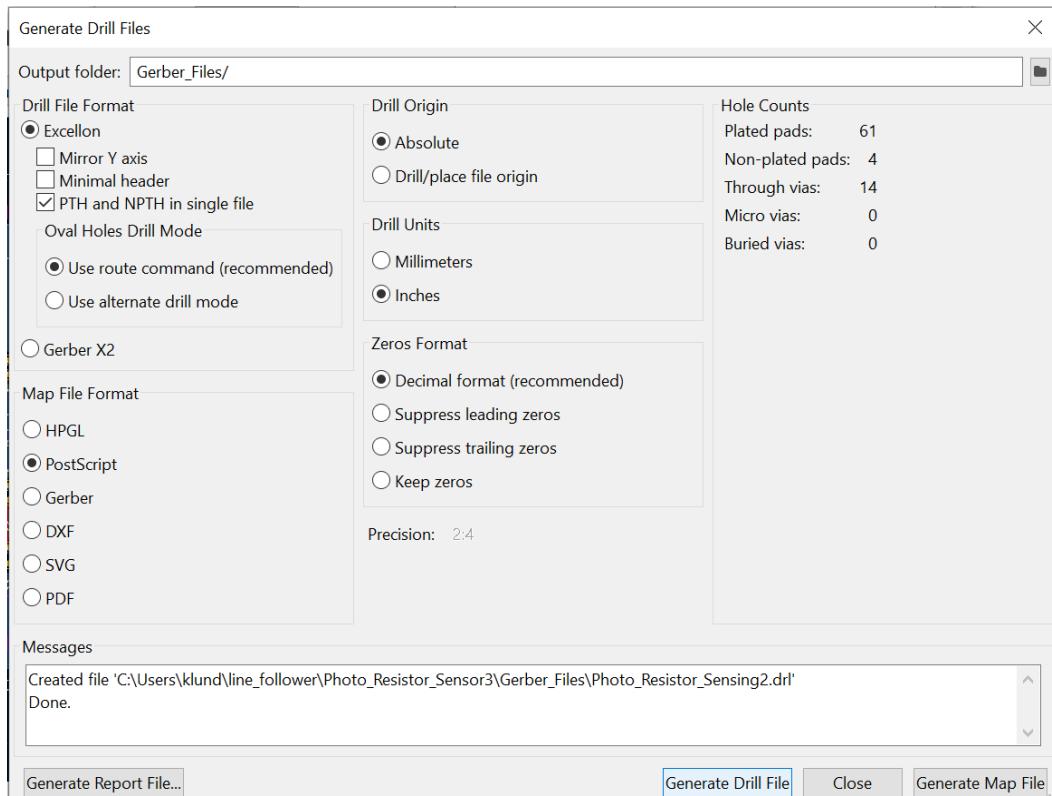
(6) Select “Generate Drill Files”. This will prompt a new window



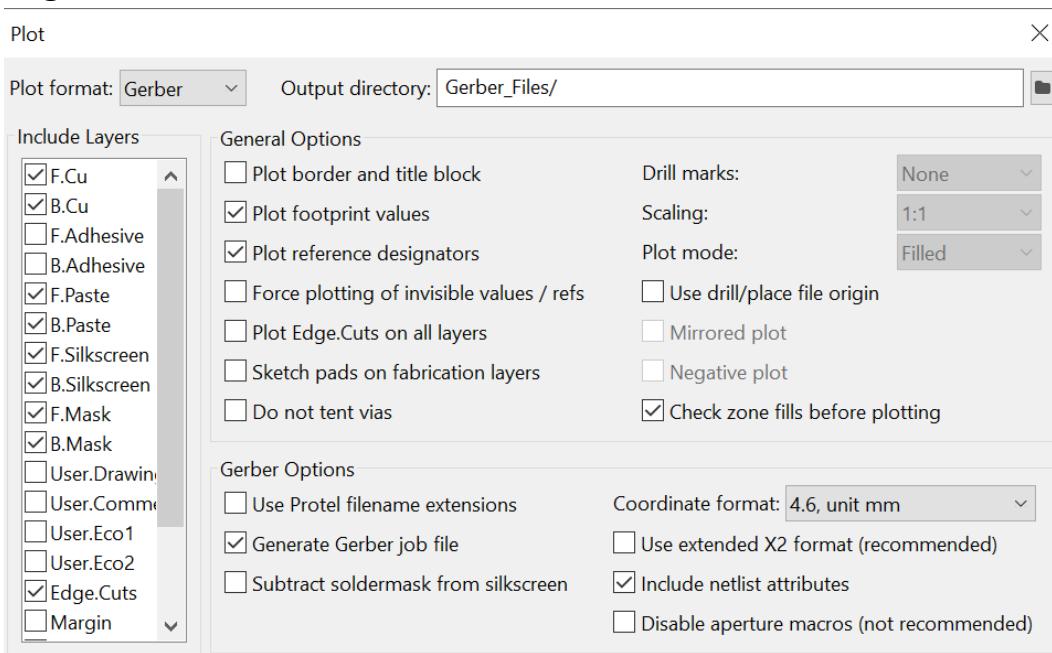
(7) Make sure the Output Folder for the Drill Files is also the folder you created. Additionally, select “PTH and NPTH in single file”, and then click “Generate Drill File”



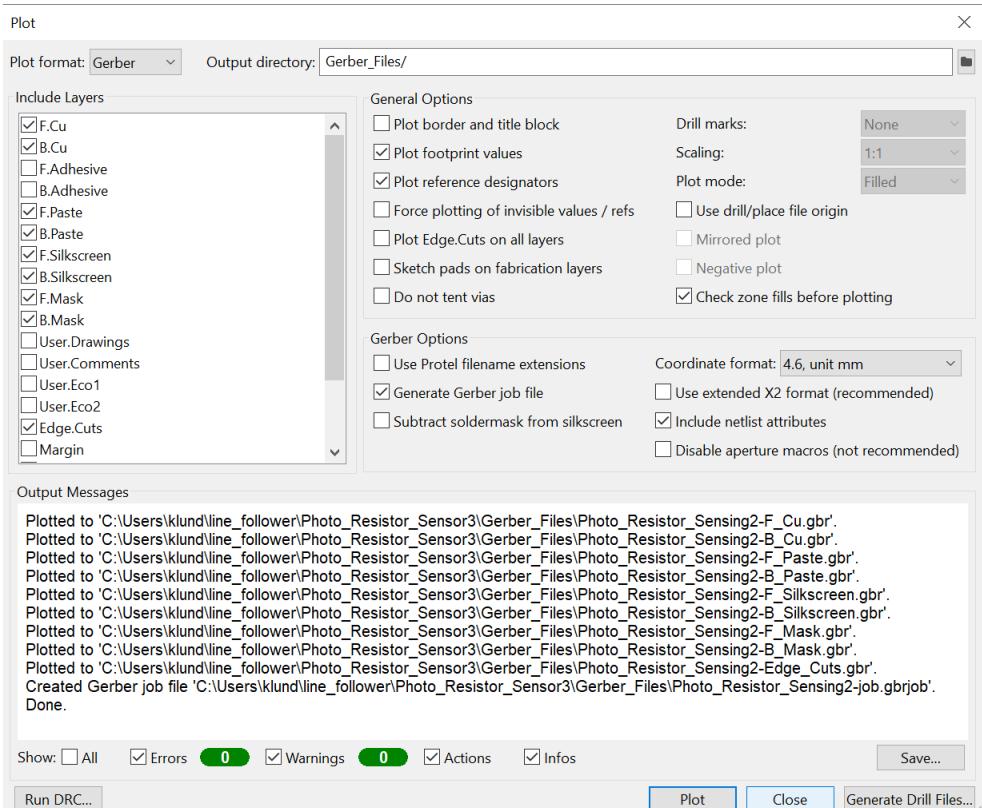
(8) You should now see a new message in the “Messages” window indicating that a .drl file was created in the specified directory. It should also say “Done”. Then, select “Close”. You should be able to see this new .drl file in the new folder



(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”.

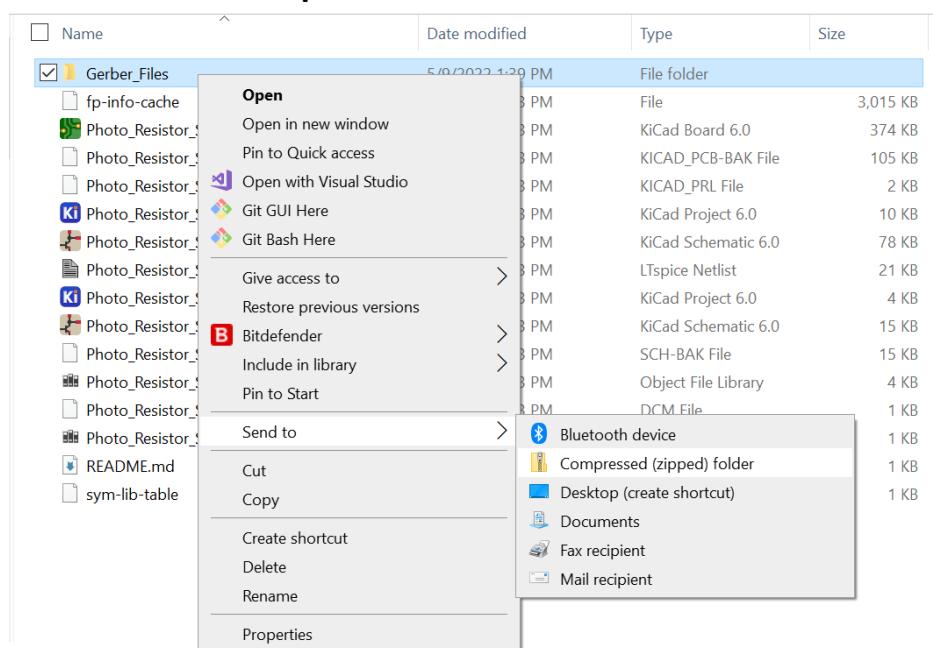


(10) Click “Plot”, and you should see one .gbr file for each layer of the PCB as well as a .gbrjob file. “Done” indicates all needed folders have been plotted



Step 3 Compress into a .zip:

(1) Next, 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

The screenshot shows the JLCPCB homepage. At the top, there's a navigation bar with links for "Live Chat", "Need Help?", "Ship to", "Why JLCPCB?", "Capabilities", "Support", "Resources", "Order now", "My file", "Sign in", and a shopping cart icon with a red '0'.

A prominent banner in the center says "Global Parts Sourcing available now" with a "New" badge. Below it, text reads "Pre-order parts from Digikey, Mouser, etc for your PCB assembly". A "Learn More >" button is present. To the right is an image of a green printed circuit board (PCB) with various electronic components like resistors, capacitors, and a microcontroller, with a blue globe graphic above it.

Below the banner are input fields for "Add gerber file" (with an "OR" option), "Layers" (set to 2), "Dimensions" (100 x 100 mm), "Quantity" (5), and a large blue "Instant Quote" button.

Step 1 Website and Account Creation

(1) Go to 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 reCAPTCHA
Privacy · Terms

Remember me [Forgot password?](#)

[Sign in with Google](#)

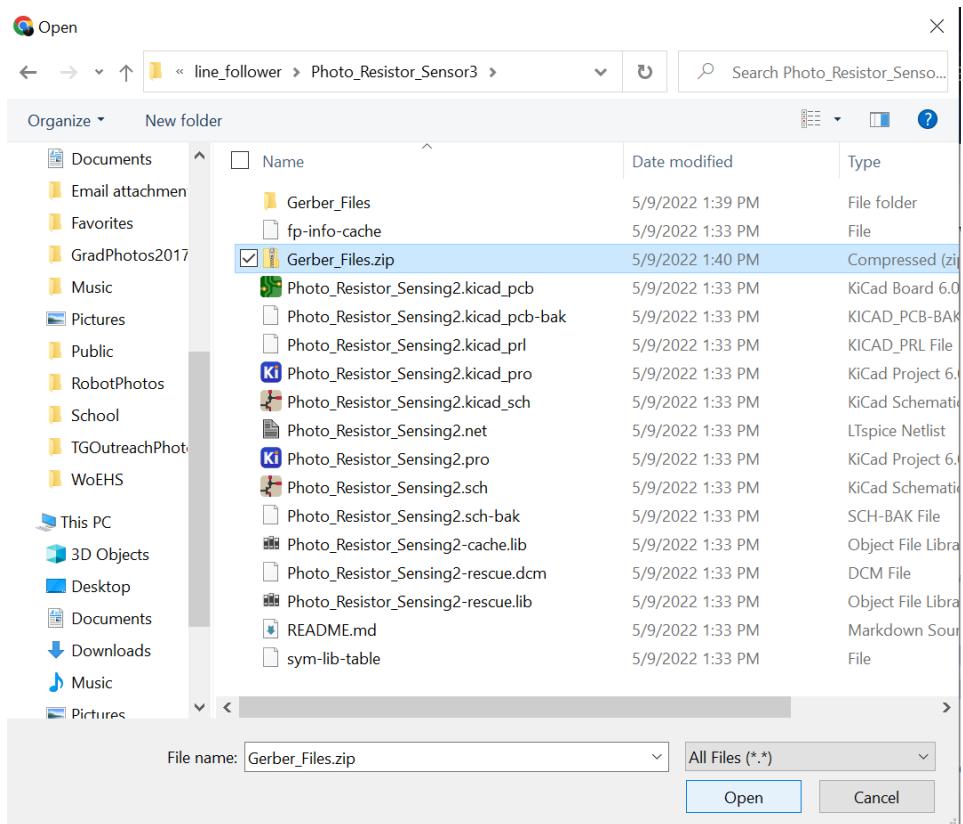
(2) Once you have logged into your new account, click “Order now” in the top right corner of the screen. You will be redirected to a page with a tab labeled “PCB”

The screenshot shows the JLCPCB website homepage. At the top, there are navigation links for "Live Chat", "Need Help?", "Ship to", "Why JLCPCB?", "Capabilities", "Support", and "Resources". Below the header, there is a promotional banner for "Global Parts Sourcing available now". The banner features a green printed circuit board (PCB) with various electronic components like resistors, capacitors, and a microcontroller, along with a blue globe. A "Learn More >" button is visible. On the left side of the main content area, there is a "Add gerber file" input field with a dashed border. To its right, there are dropdown menus for "Layers" (set to 2), "Dimensions" (100 x 100 mm), and "Quantity" (5). A large blue "Instant Quote" button is positioned to the right of these fields.

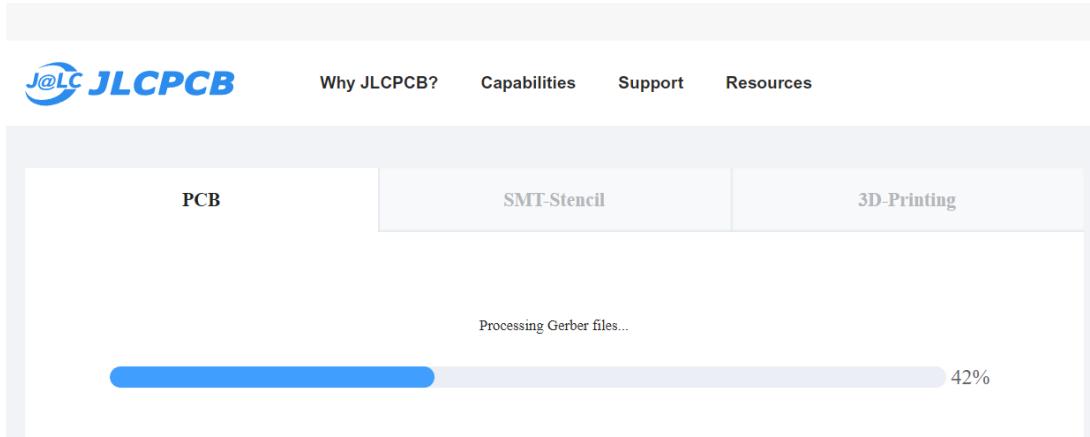
(3) Next, select “Add gerber file” which will prompt you to upload a .zip file

The screenshot shows the JLCPCB PCB ordering page. At the top, there are navigation links for "Live Chat", "Need Help?", "Ship to", "Why JLCPCB?", "Capabilities", "Support", and "Resources". Below the header, there are three tabs: "PCB" (selected), "SMT-Stencil", and "3D-Printing". The main form area has sections for "Base Material" (FR-4 selected), "Layers" (2 selected), and "Dimensions" (100 x 100 mm). A prominent blue "Add gerber file" button is located in a red-bordered box. Below it, there is a note: "Only accept zip or ruc, Max 10 M, View example >". There are also links for "Instructions for ordering" and "Log in to view your upload history". To the right, there is a "Charge Details" sidebar with "Special Offer" (\$2.00), "Build Time" (1-2 days), and "Calculated Price" (\$4.00 - \$2.00). A "SAVE TO CART" button is at the bottom of the sidebar. A "Shipping Estimate" section and a "Charge: Choose destination country first" link are also present.

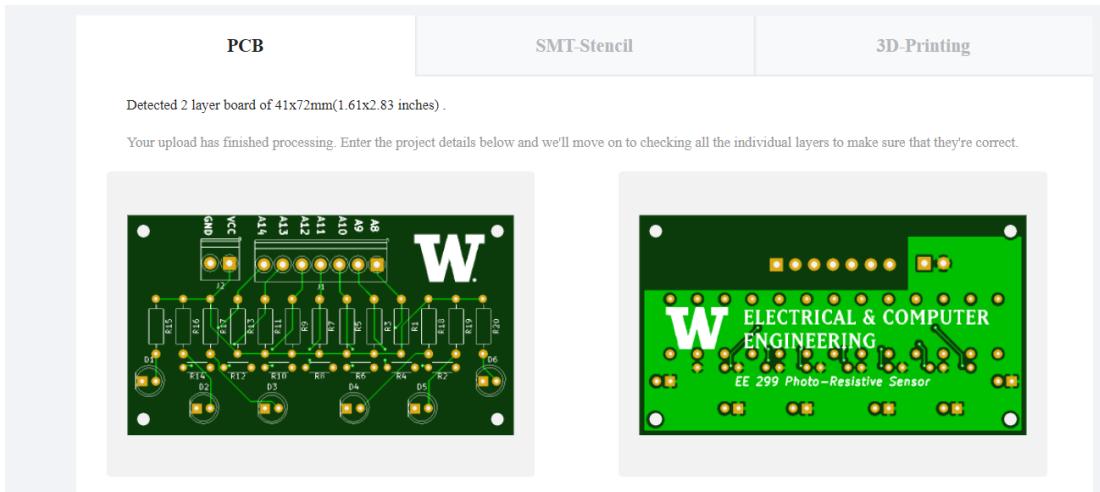
(4) Select your compressed Gerber_Files.zip folder and click “Open”



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	<input checked="" type="radio"/> FR-4	<input type="radio"/> Aluminum							
Layers	<input checked="" type="radio"/> 1	<input checked="" type="radio"/> 2	<input type="radio"/> 4	<input type="radio"/> 6					
Dimensions	<input checked="" type="radio"/> 41	*	<input checked="" type="radio"/> 72	mm <input type="radio"/>					
PCB Qty	<input checked="" type="radio"/> 5								
Product Type	<input checked="" type="radio"/> Industrial/Consumer electronics	<input type="radio"/> Military/Aerospace	<input type="radio"/> Medical						
Different Design	<input checked="" type="radio"/> 1	<input type="radio"/> 2	<input type="radio"/> 3	<input type="radio"/> 4	<input type="radio"/>				
Delivery Format	<input checked="" type="radio"/> Single PCB	<input type="radio"/> Panel by Customer	<input type="radio"/> Panel by JLCPCB						
PCB Thickness	<input type="radio"/> 0.4	<input type="radio"/> 0.6	<input type="radio"/> 0.8	<input type="radio"/> 1.0	<input checked="" type="radio"/> 1.6	<input type="radio"/> 2.0			
PCB Color	<input checked="" type="radio"/> Green	<input type="radio"/> Purple	<input type="radio"/> Red	<input type="radio"/> Yellow	<input type="radio"/> Blue	<input type="radio"/> White	<input checked="" type="radio"/> Black		
Silkscreen	<input checked="" type="radio"/> White								
Surface Finish	<input type="radio"/> HASL(with lead)	<input checked="" type="radio"/> LeadFree HASL-RoHS	<input type="radio"/> ENIG-RoHS						
Outer Copper Weight	<input checked="" type="radio"/> 1 oz	<input type="radio"/> 2 oz							
Gold Fingers	<input checked="" type="radio"/> No	<input type="radio"/> Yes							
Confirm Production file	<input checked="" type="radio"/> No	<input type="radio"/> Yes							
Flying Probe Test	<input checked="" type="radio"/> Fully Test	<input type="radio"/> Not Test							
Castellated Holes	<input checked="" type="radio"/> No	<input type="radio"/> Yes							
Remove Order Number	<input checked="" type="radio"/> No	<input type="radio"/> Yes	<input type="radio"/> Specify a location						
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.

Charge Details



Special Offer

\$2.00

Build Time

PCB: 1-2 days

\$0.00

Calculated Price

~~\$4.00~~ **\$2.00**

Additional charges may apply for [special cases](#)

Weight

0.16kg

SAVE TO CART

Shipping Estimate

Charge: [Choose destination country first](#)

(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!