Description
This document is a guide to designing a PCBA prototype in Kicad and ordering it through JLCPCB assembly in a way that is quick and easy.

Pre-Requisites and links
Kicad - Open Source EDA software
JLCPCB - Online PCB fabrication website
JLC Parts Search - third party tool for better searching of available SMD assembly parts (github)
Part import - third party tool for importing JCL component into Kicad Library from command line
JLC Export - third party tool for exporting BOM and CPL data

Adding JLC compatible parts to Kicad
Search desired components using JLC Parts Search to find the LCSC number

Components matching query: 69 — wget all datasheets

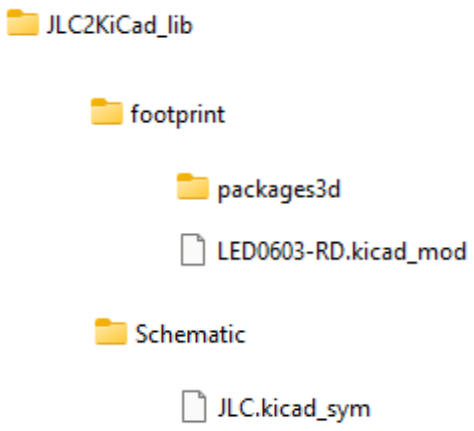
MFR (click for datasheet)	LCSC	Basic/Extended	Image	Description	Manufacturer	Stock	Price
SZYY0603R	C434419	E		Colorless transparence -40℃~+85℃ 615nm~630nm Red 120° 40mW 0603 Light Emitting Diodes (LED) ROHS	Yongyu Photoelectric	1963640	0.008$/unit 0.008$/1 units
NCD0603R1	C84263	E		Red 0603 Light Emitting Diodes (LED) ROHS	Foshan NationStar Optoelectronics	1688116	0.008$/unit 0.008$/1 units
XL-1608SURC-06	C965799	E		Colorless transparence -30℃~+85℃ Positive post 617nm~621nm Red 120° 55mW 0603 Light Emitting Diodes (LED) ROHS	XINGLIGHT	1164563	0.003$/unit 0.003$/1 units
19-217/R6C-AL1M2VY/3T	C72044	E		Colorless transparence -40℃~+85℃ 617.5nm~633.5nm Red 120° 60mW 0603 Light Emitting Diodes (LED) ROHS	Everlight Elec	944439	0.015$/unit 0.015$/1 units

Import components using Part import.

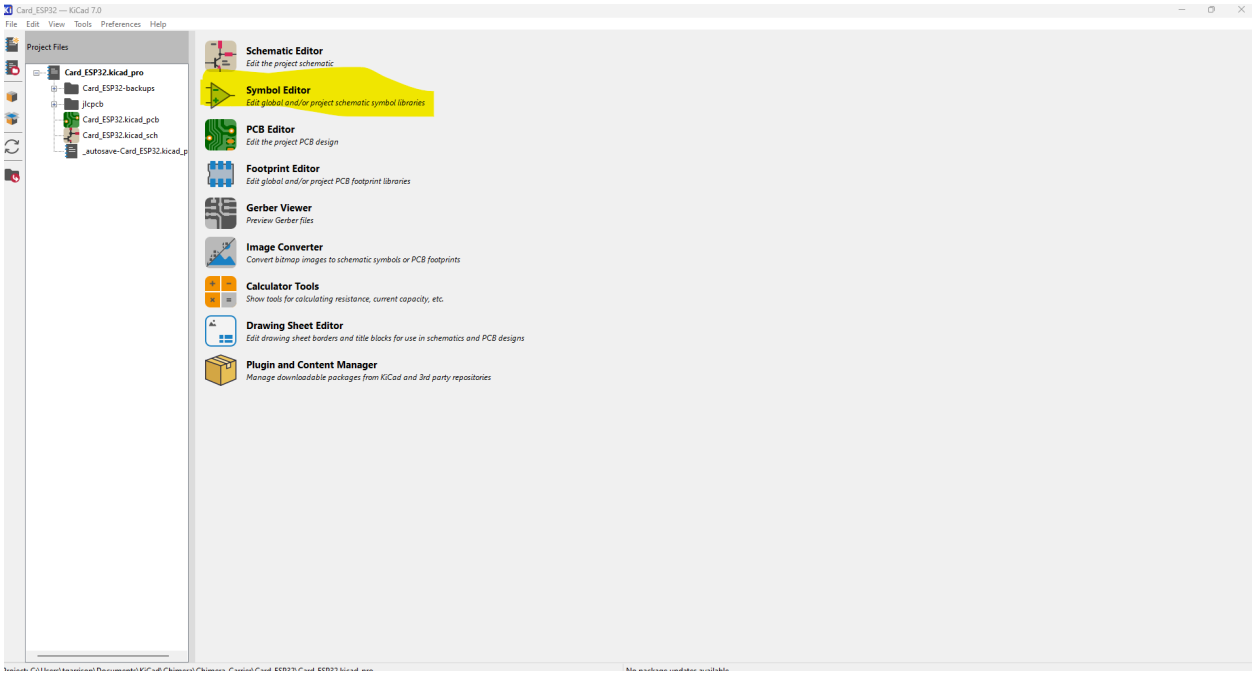
- Open terminal session in desired directory (e.g. *JLC_libraries*)
- In the terminal, type command

```
JLC2KiCadLib C72044 -schematic_lib JLC
```

- The following folders will be created

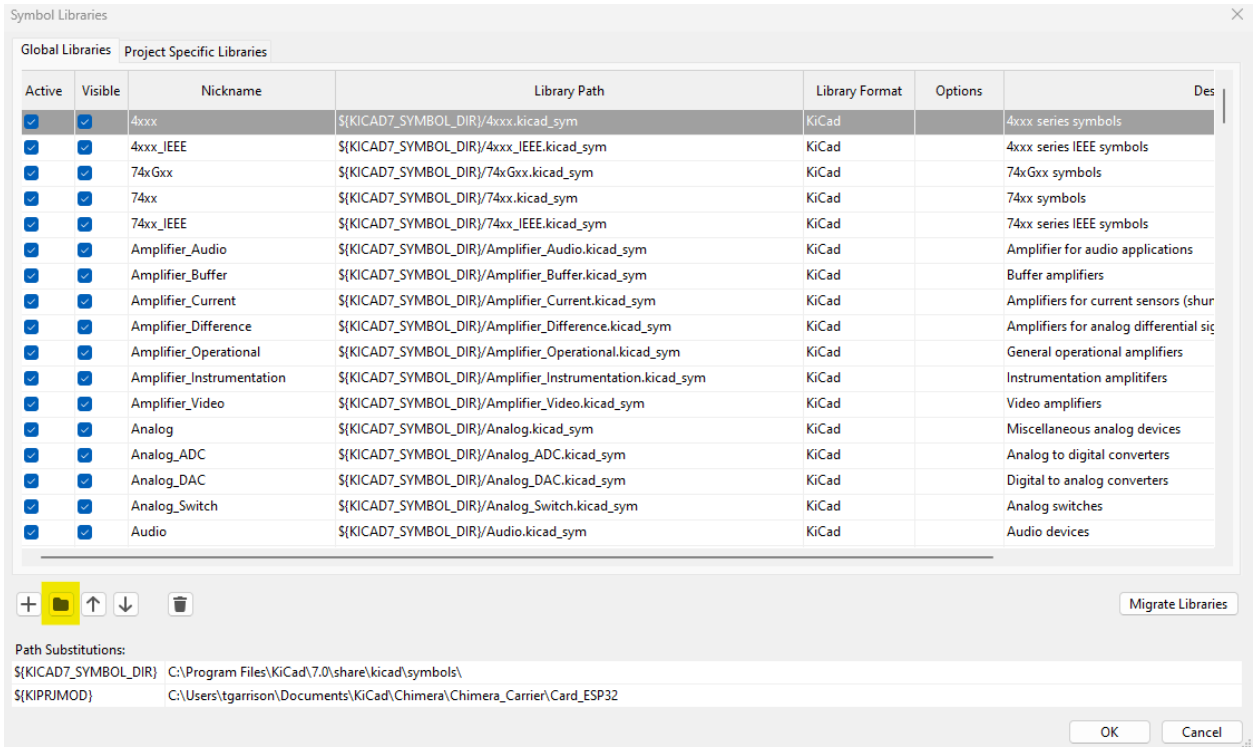


- Open KiCAD and open the symbol editor



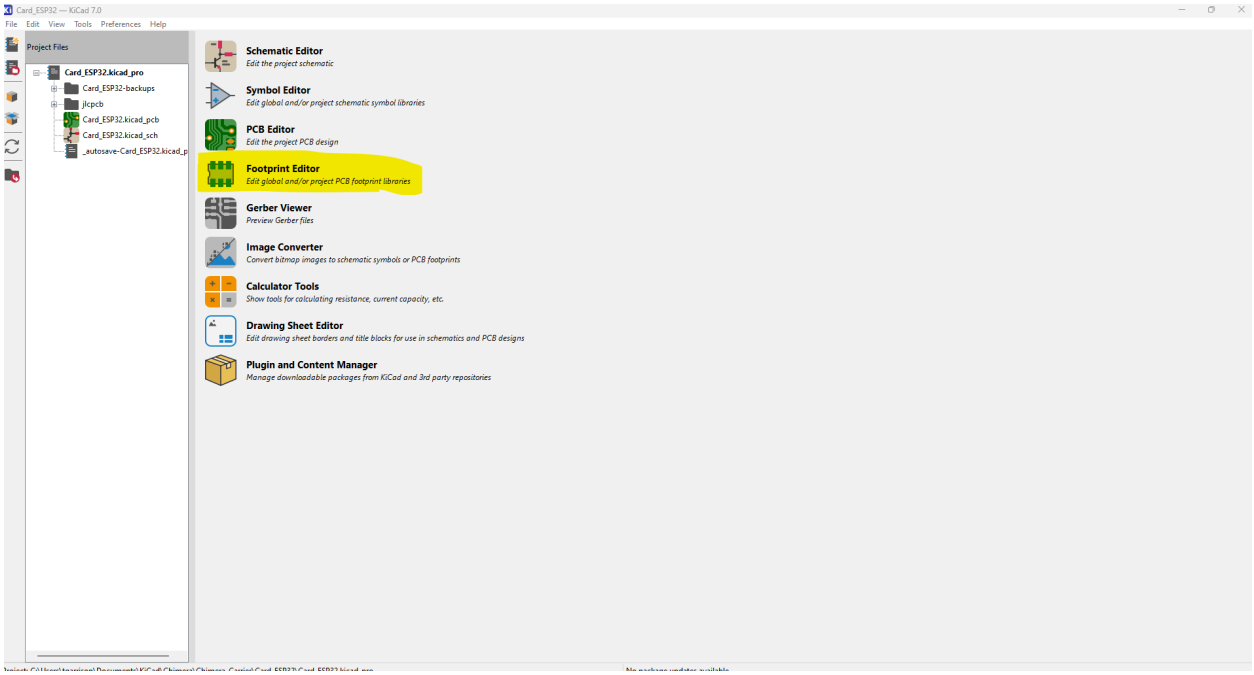
- Go to *Preferences* > *Manage Symbol Libraries…*

- Press add existing

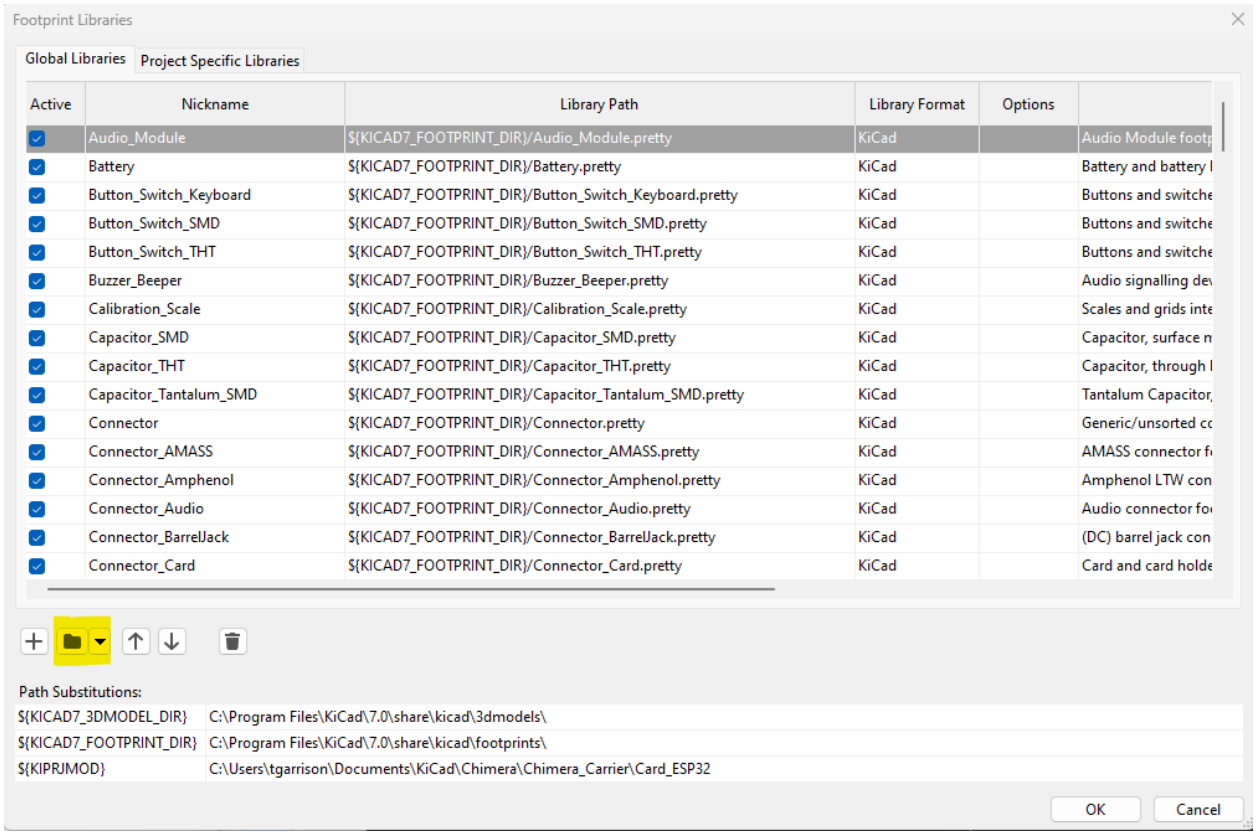


- Browse to the JLC.kicad_sym file that was created in the Schematic folder
- Press ok

- Close the symbol editor and open the footprint editor

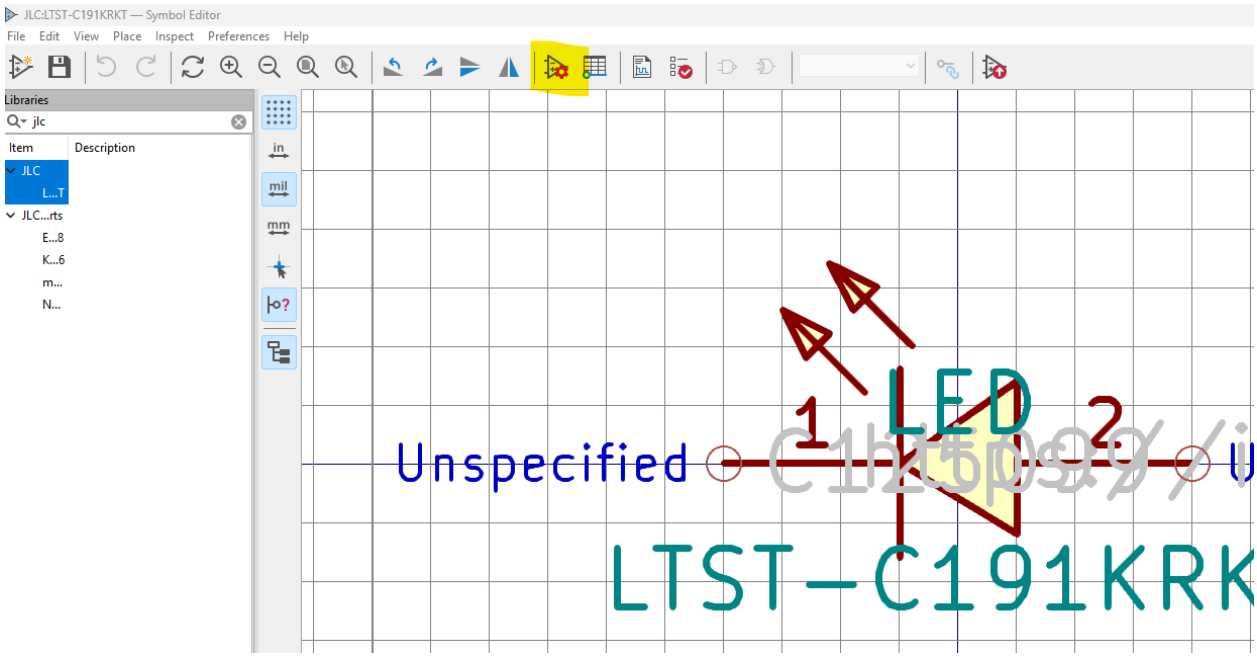


- Go to *Preferences > Manage Footprint Libraries…*
- Press Add existing

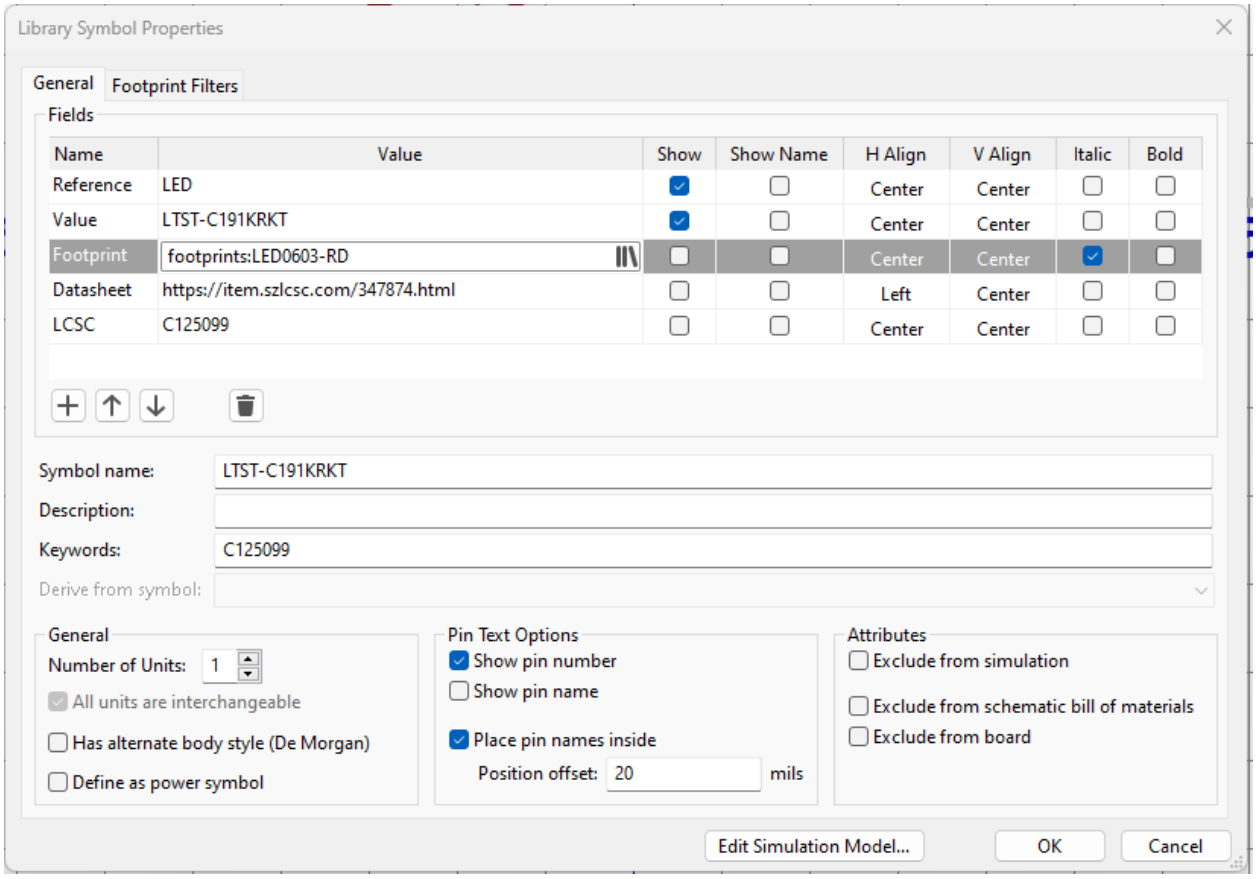


- Select the *footprint* folder that was created (optionally you can give it a nickname)
- Press okay

- To assign the symbol to the footprint, open the symbol editor again
- Select the recently created component/s and open the symbol properties dialogue



- Click the icon in the footprint field and select the correct footprint from the previously imported footprint library



- This will permanently associate the part with the footprint.

- This step can be eliminated if the created footprint is moved to the Kicad Program files and placed in a footprint.pretty folder .\KiCad\7.0\share\kicad\footprint.pretty (Untested)

- You can now add the part to the schematic with the confidence that when you go to order a prototype from JLCPCB it will automatically be able to be found and assembled when you import BOM and CPL files.

Exporting files for JLC

- After finishing your Kicad Design and are ready to order, you can now press the JLC icon that was added to Kicad after you added the JLC Export tool to Kicad.
- Here you can review that all of your parts are recognized as JLC assembly components by their CXXXXX number.
- Press Generate fabrication files and zip them into a folder to drag into JLCPCB when ready to order. This should let you order the prototype with no further work unless one of the parts you used has gone out of stock.