

How to import Makerstock KiCad component libraries

Written By: Matt P. Lamparter



Step 1 — Notes on where to save libraries



- Before we begin it is important to note that if you are using a public computer (any computer that is not your own personal computer) then you should not store these library files locally.
- Storing files locally means saving them to the hard drive of the computer you are working on (in locations like the Desktop or the Downloads folder).
- Storing files locally means that if you save your work and then attempt to open it on a different computer you will not be able to access the libraries saved on the previous computer.
- The correct location to save libraries is on networked storage such as Google Drive or Bucknell's Netspace. Libraries saved in a networked location can always be accessed from any computer.
- In the following steps, please be sure to save all library files to networked storage.

Step 2 — Download necessary files

← →	C https://drive.google.com/drive/folders/1-YSZVE_AS1Sup	pvm5uDBSZUKPvrUfkS8	Image: Share of the state o	9905 ra large icons El Large icons Se Medium all icons Se List Elist Elist Elist rs Elist Content	icons	Group by *	☐ Item check boxes ✔ File name extensi ✔ Hidden item:
KiCad			Panes	Layout		Current view	Show/I
	Name 1	Owner	$\leftarrow \rightarrow \sim 1$ > This PC >	> Desktop > Makerstock20230905 >			
1	Makerstock.3Dshapes	Owner hidden	Name	Date modified	Type File folder	Size	
l	Makerstock.pretty	Owner hidden	Makerstock.pretty	9/5/2023 9:14 AM	File folder		
	Makerstock.kicad_sym 🐣	Owner hidden	Makerstock.kicad_sym	9/5/2023 9:14 AM	KICAD_SYM File	424 KB	
l l	₩ Makerstock20230905.zip	Owner hidden					

- Visit the public Google Drive folder where the library files are stored.
- Download the MakerstockYYYYMMDD.zip file and note where you save it.
- Unzip the file and you should have a single file and two folders as shown here in the second image.
- (i) Note that local storage (the Desktop) is used in this example. Please do not save things to local storage as you complete this tutorial!

Step 3 — Import symbols library

- Launch KiCad and open the Symbol Editor.
- Select File -> Add Library from the menu bar at the top. Navigate to the location where you
 unzipped the .zip file you downloaded and select the Makerstock.kicad_sym file.
- (i) Depending on the settings for your operating system, you may not see the .kicad_sym extension.
- Confirm that you now see the Makerstock symbols library in the left column of libraries. Try double clicking on an arbitrary symbol within the library to confirm that the symbol is loaded on the right side of the screen.
- When you are confident the Makerstock symbols library has loaded properly you can exit the Symbol Editor.

Step 4 — Import footprints library

Image: The grad grad grad grad grad grad grad grad	Ino footprint leaded) Footprint Editor File Edit View Pice Inspect Tools Preferences Help Image: Security Pice Inspect Tools Pice Inspect Pice Inspect Tools Pice Inspect Pice Pice Pice Pice Pice Pice Pice Pice	Materiolock4122M - Fortprint Editor Tel Edit Yore Base Juscet Tools Interences Igins
	Vscillator Sillator Sillator	318239105

- Now launch the Footprint Editor.
- Select File -> Add Library from the menu bar at the top. Navigate to the location where you
 unzipped the .zip file you downloaded and select the Makerstock.pretty folder.
- Confirm that you now see the Makerstock footprint library in the left column of libraries. Try double clicking on an arbitrary footprint within the library to confirm that the footprint is loaded on the right side of the screen.
- When you are confident the Makerstock footprint library has loaded properly you can exit the Footprint Editor.
- (i) There is one additional optional step that may be completed next, prior to exiting the Footprint Editor.

Step 5 — Confirm 3D symbols (OPTIONAL)



- While this step is not required for designing PCBs, it is helpful to be able to render a populated PCB in 3D to view what the finished product will look like.
- With the Footprint Editor window still open, locate a footprint with a known 3D model included. In this example we will use the **12401598E4#2A** USB C connector. Double click the footprint to load it on the right side of the window.
- With your mouse hovering over the black area on the right side of the screen press the **E** key on your keyboard to display the **Properties** dialog window.
- If the 3D models were properly loaded with the footprint library you should see a rendering of a USB C connector. If you do not see any 3D rendering please contact the Lab Director for assistance in loading these models.