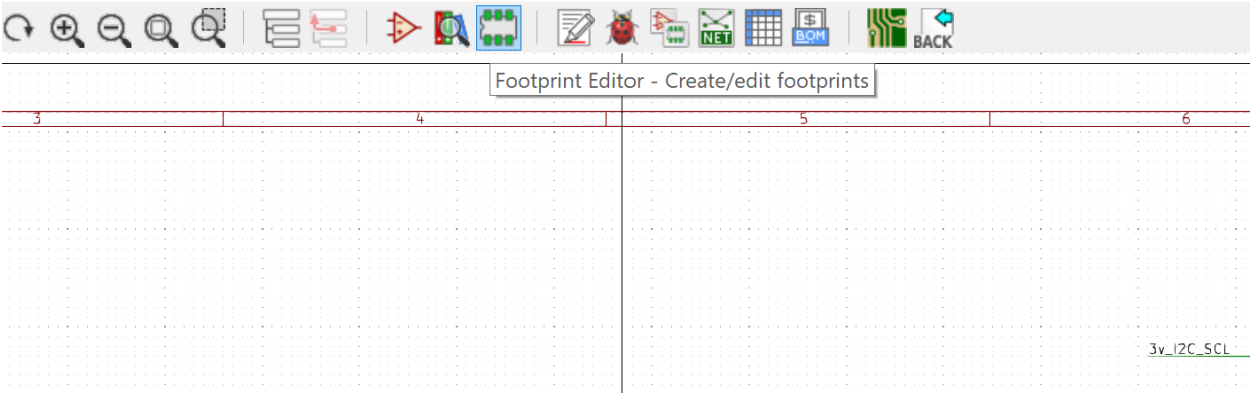
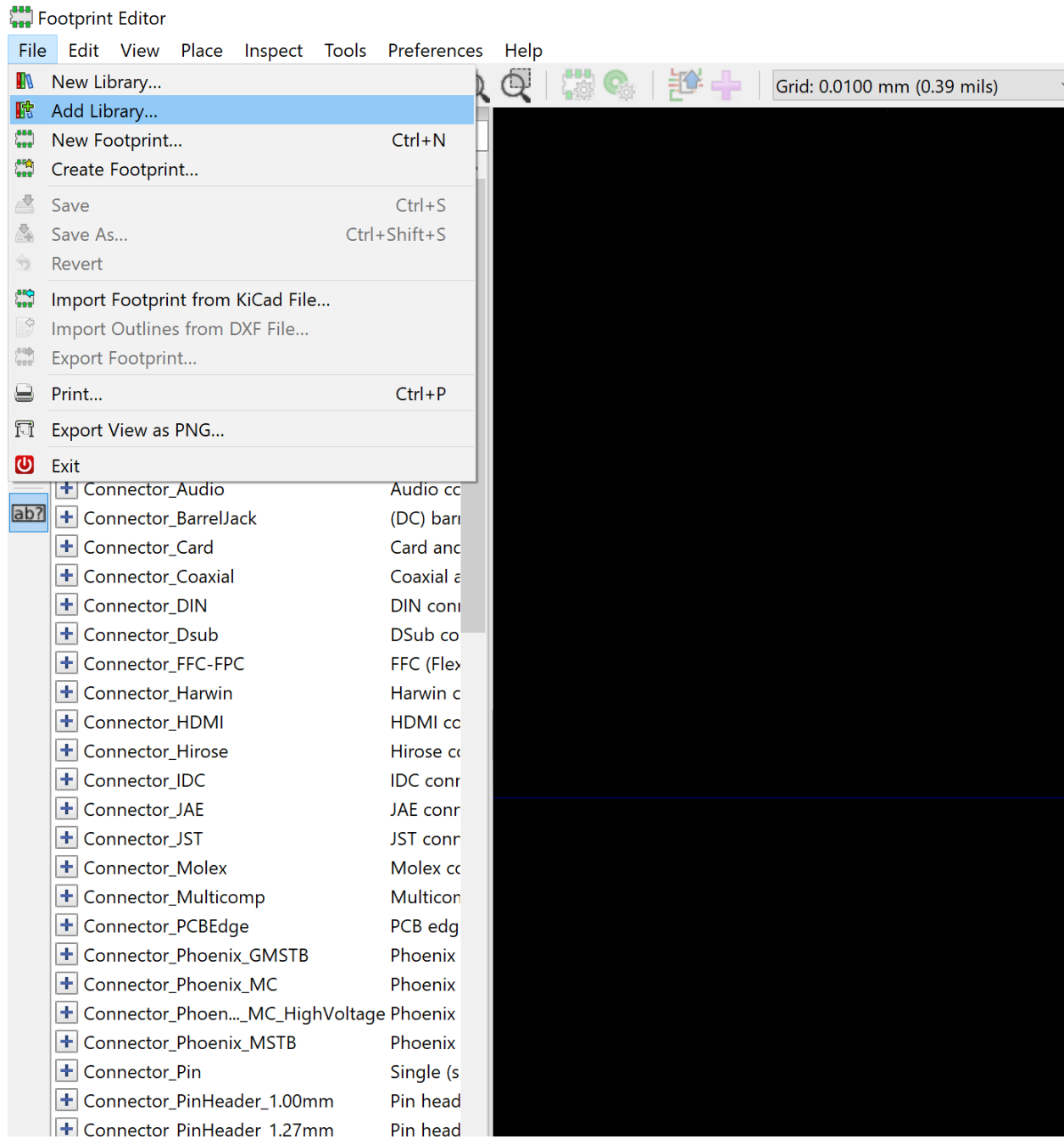


First you'll need to add the footprint library to the project

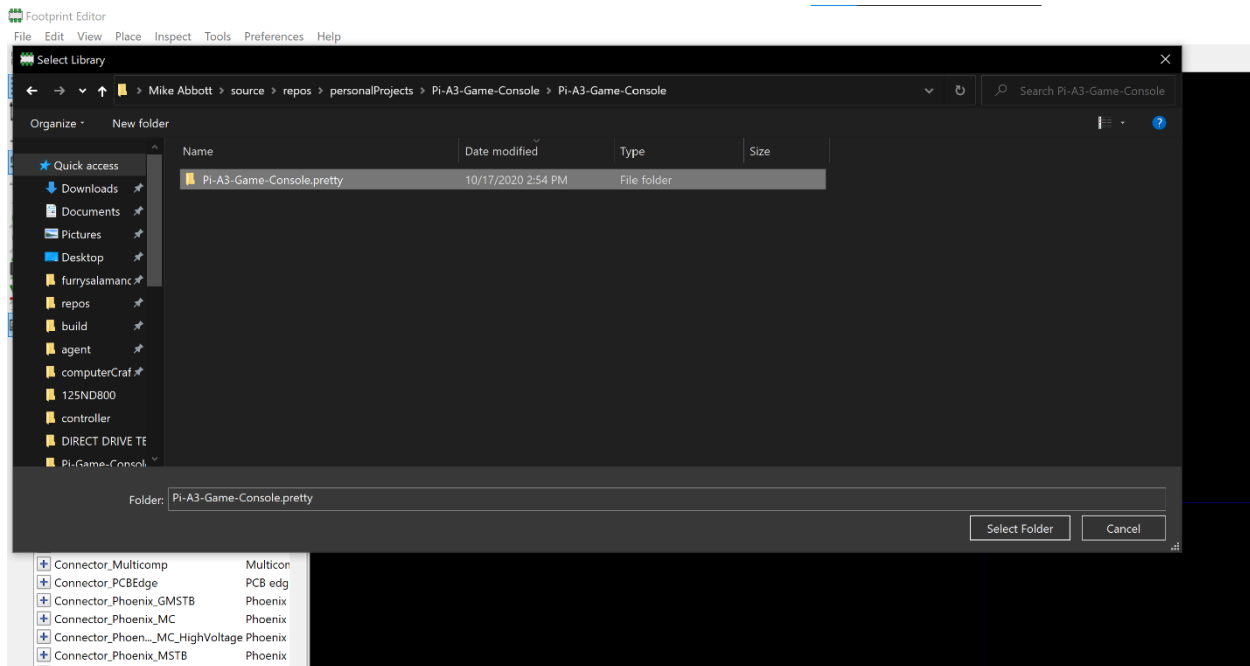
source\repos\personalProjects\Pi-A3-Game-Console\Pi-A3-Game-Console

o

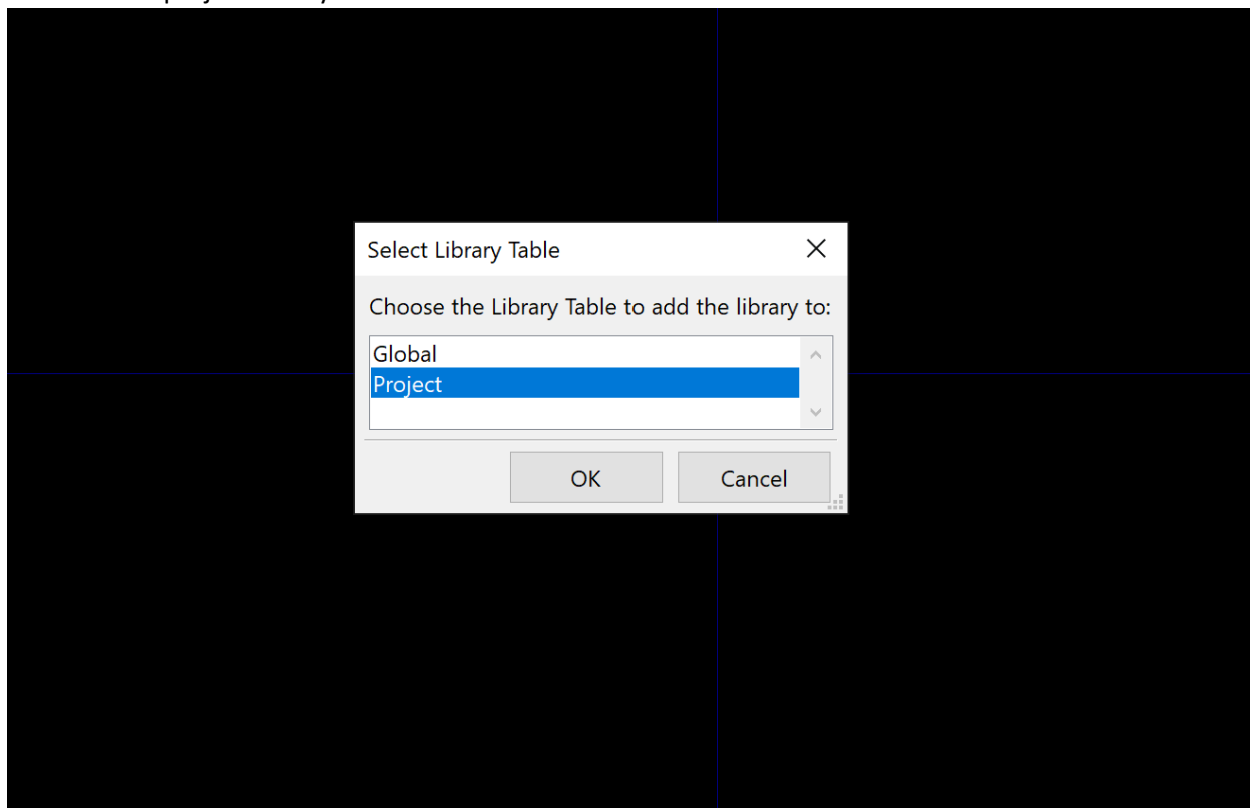




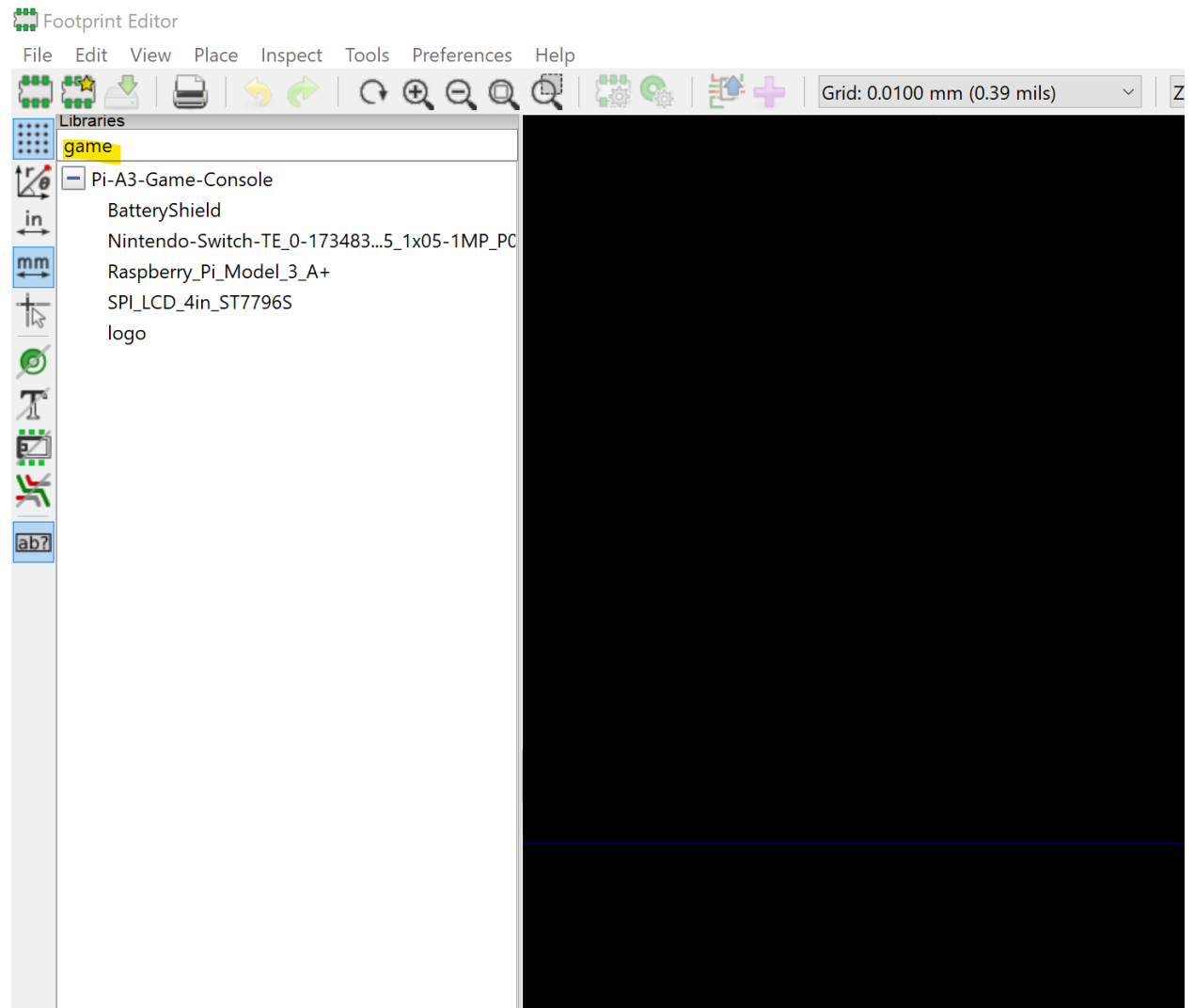
Locate the folder included with this zip file, and copy it into your project's folder. Then, locate that folder in the select library menu, and add it like so:



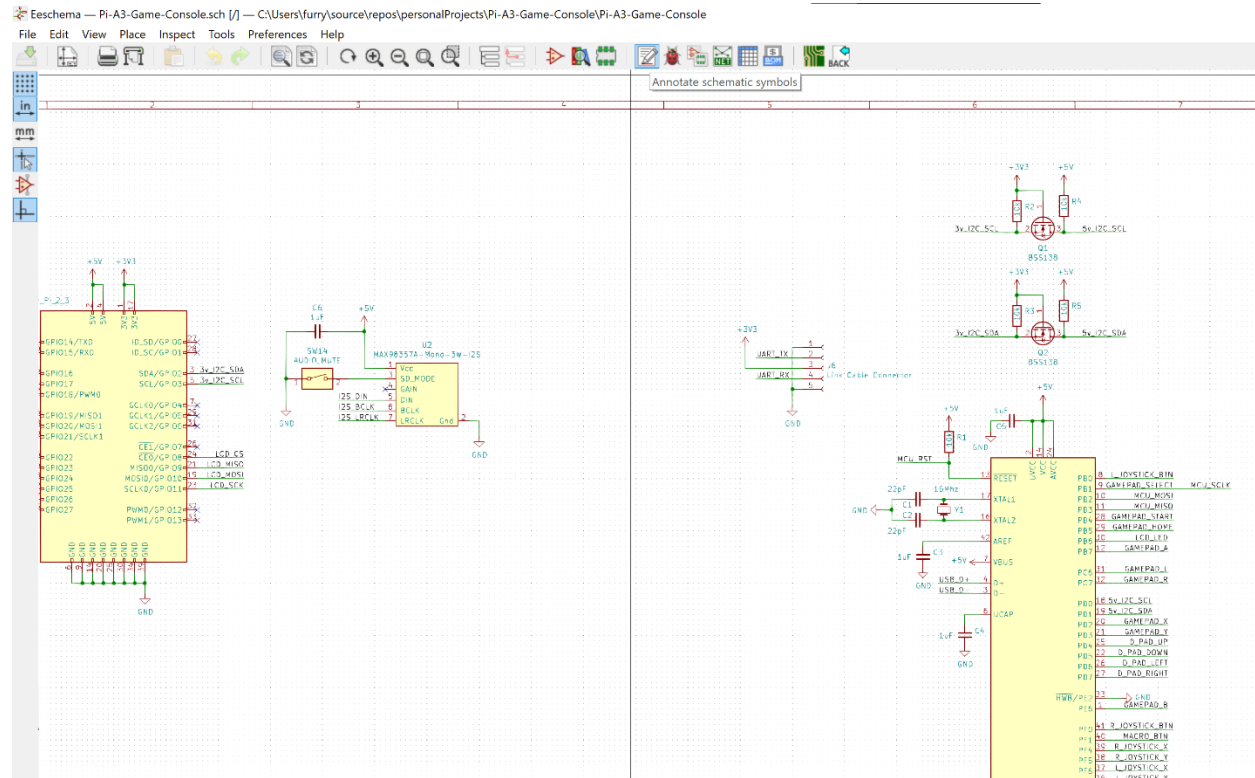
Add it to the project library table



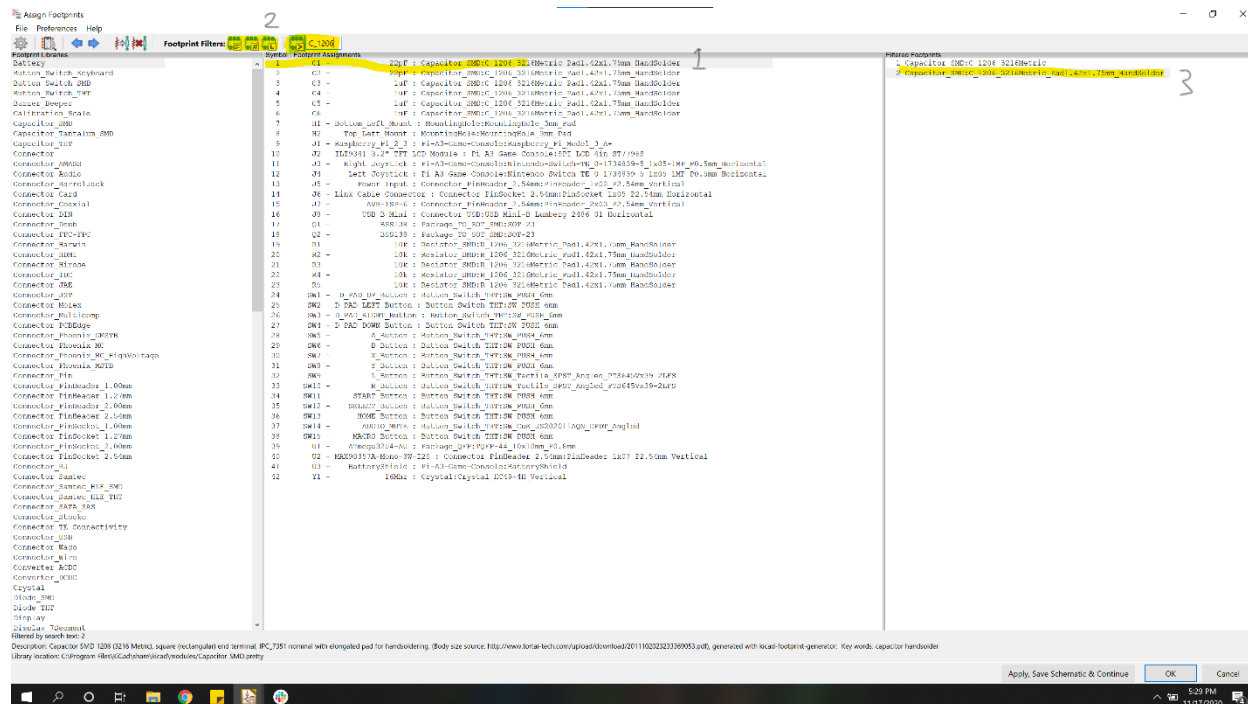
Verify that it was successfully added by searching for it in the footprint editor. If it's there, you can close this window and proceed to the next step.



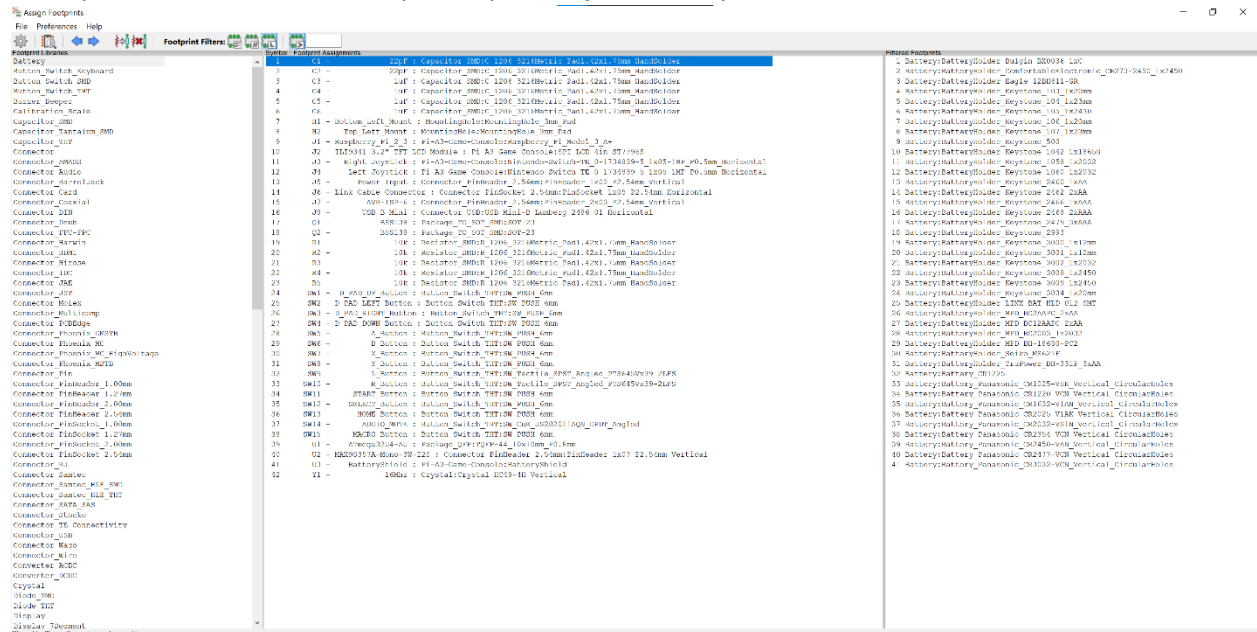
Next you'll need to assign part numbers to each component. Do this using this button.



This part is a little tricky, so I'm going to suggest the easiest way to do it. First, select the top component. Then, uncheck all three of those boxes on the left hand side of label 2. This may cause your computer to lag for a second. Then make sure the box right next to the search bar is checked. Then you can just search for the footprint names. Once you've found the matching footprint, go ahead and double click it over on the right side. That will assign that footprint to that symbol. Go through and assign footprints to each schematic symbol. You'll need to match them based on the value of each part, IE, 22pF, and *NOT* the symbol number (IE, C1) because it may be different on your project.



Once you've done this for each symbol, you can just hit okay.



Generate the netlist, accepting all the default values and settings. This is just a list of all the connections in your schematic.

jos\personalProjects\Pi-A3-Game-Console\Pi-A3-Game-Console

