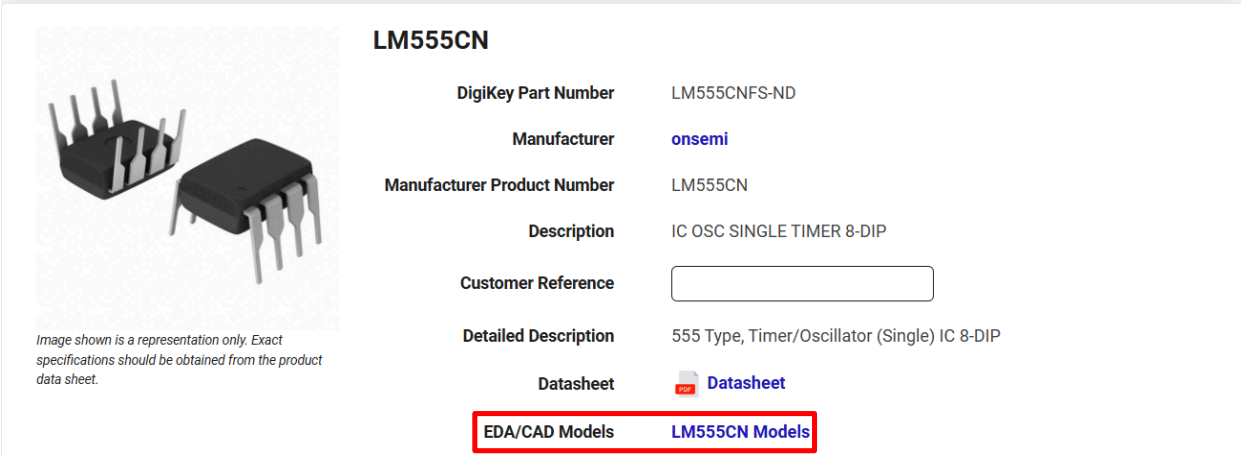
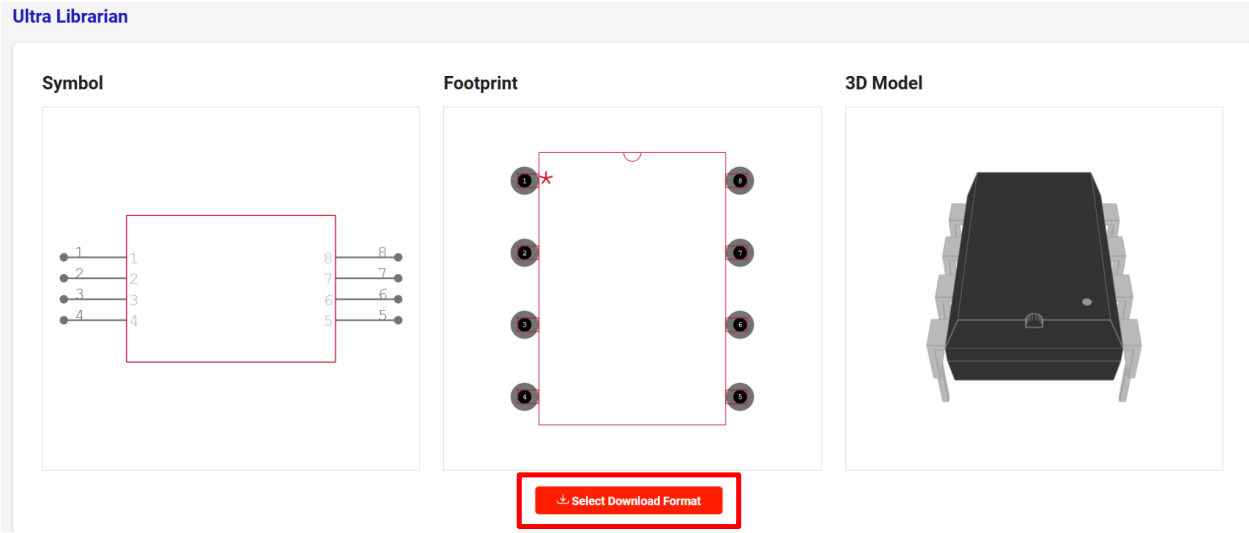
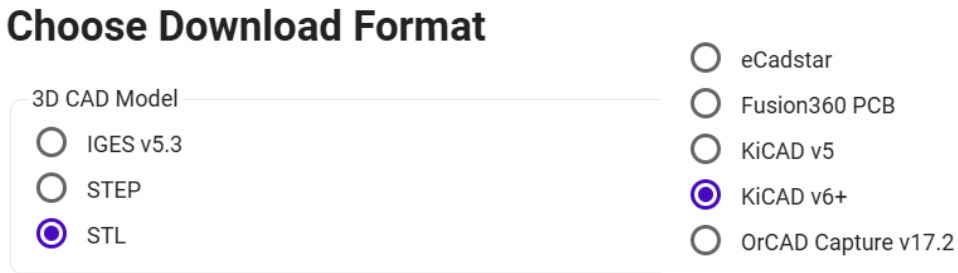
Importing Symbols

1. Find part on DigKey

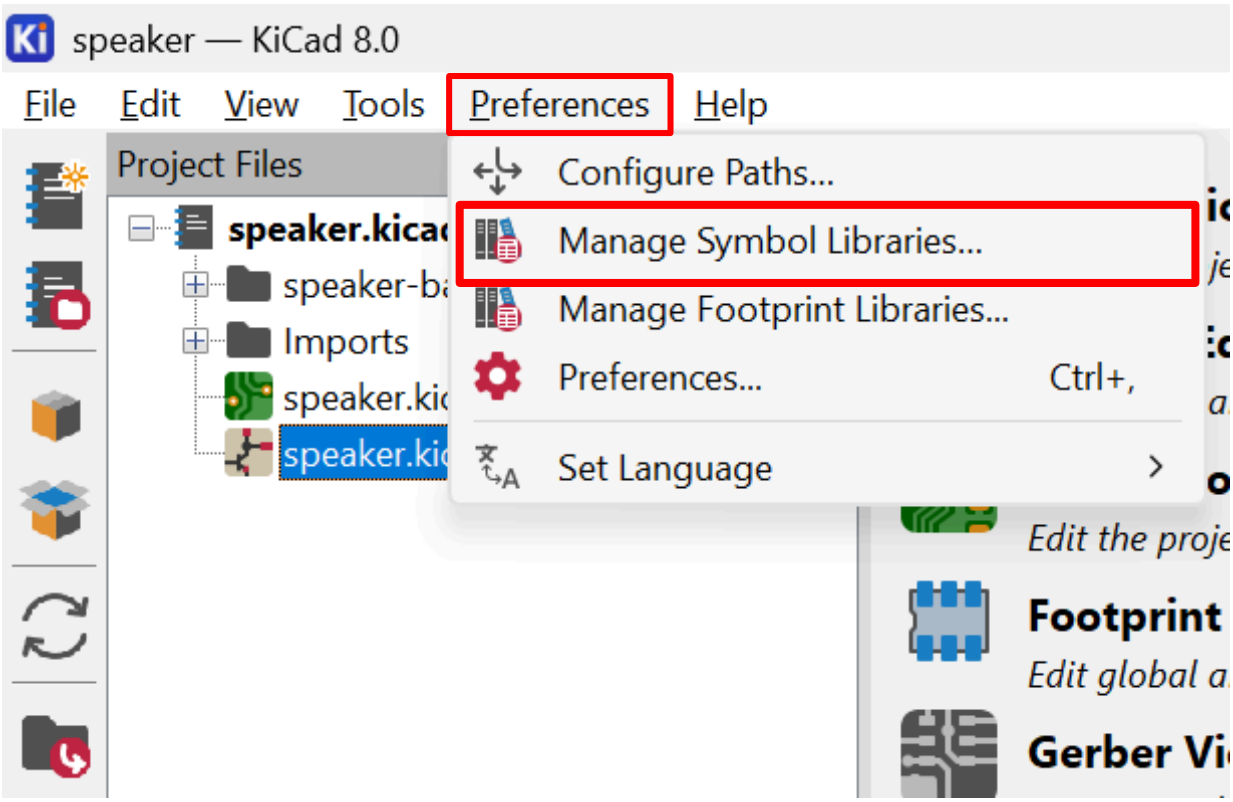


2. Navigate to EDA/CAD Models
3. Preview EDA/CAD models for accuracy to part

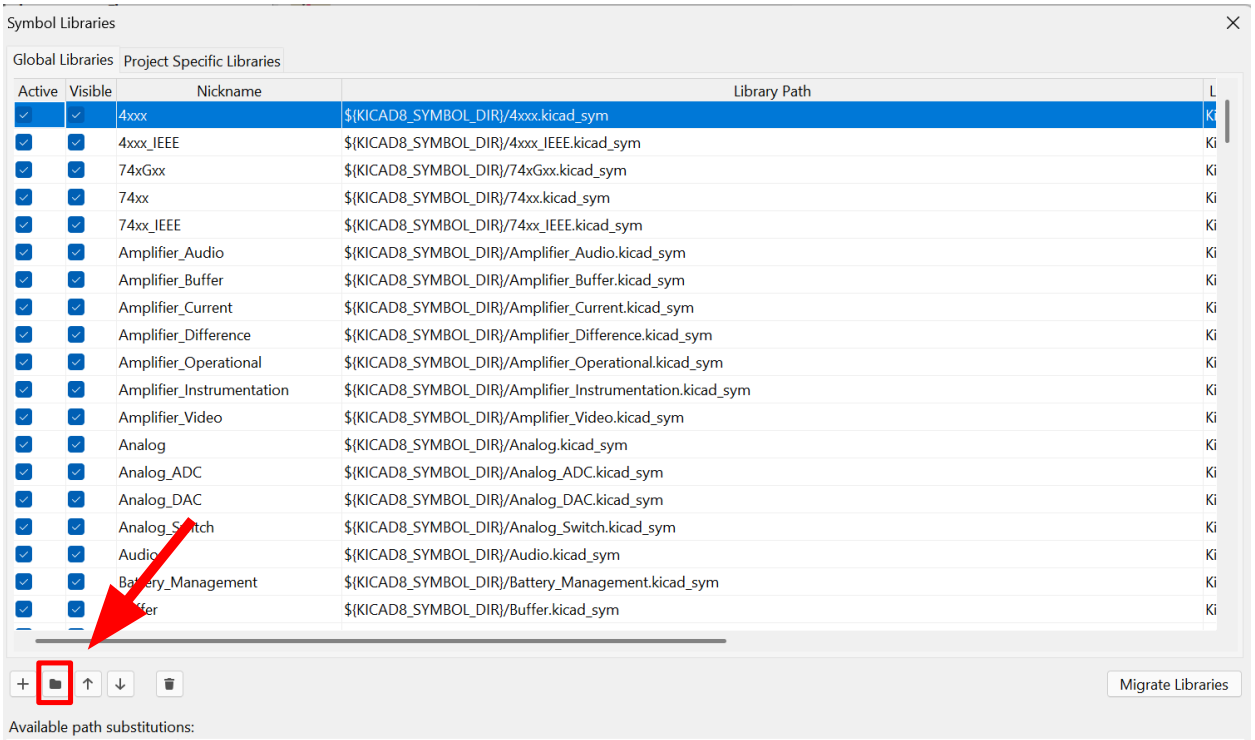




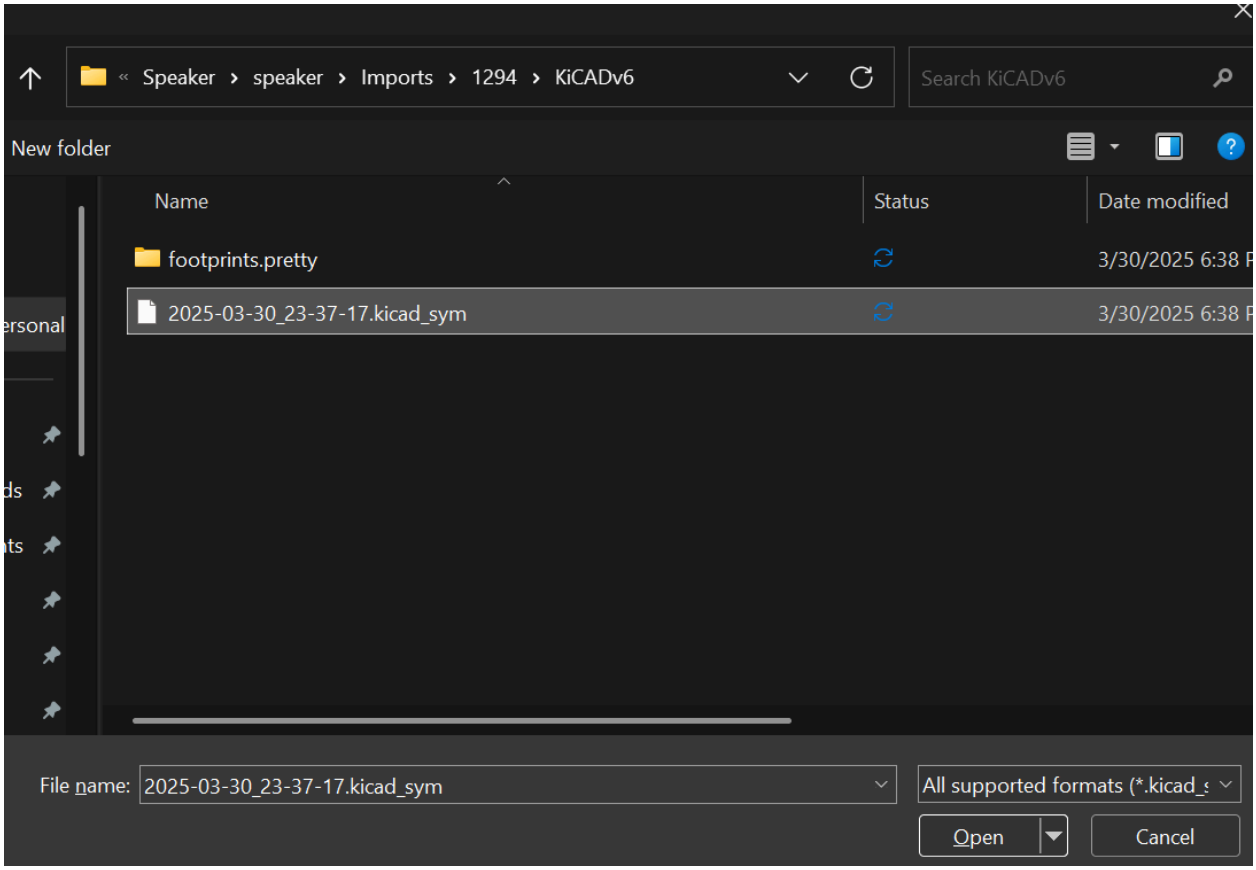
4. Choose download format (KiCad v6+)
5. Unzip into desired file location making sure to take note of the file path



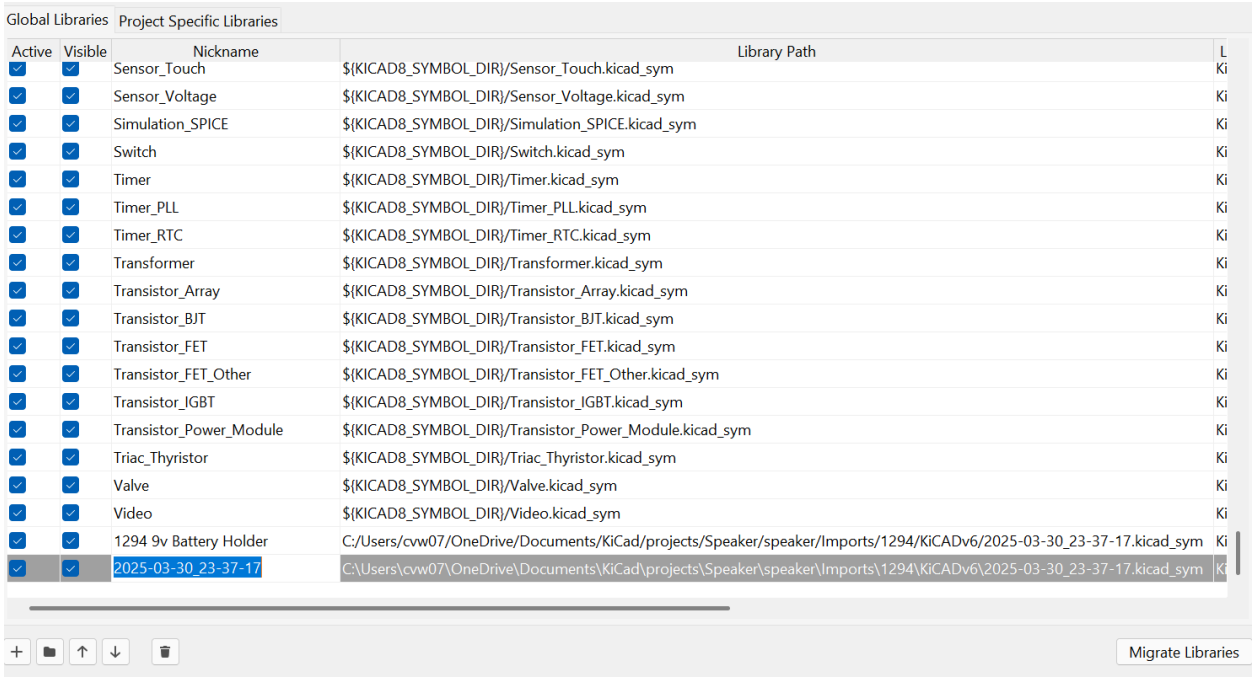
6. To import the symbol open the current version of KiCad installed on your device and navigate to “Manage Symbol Libraries”



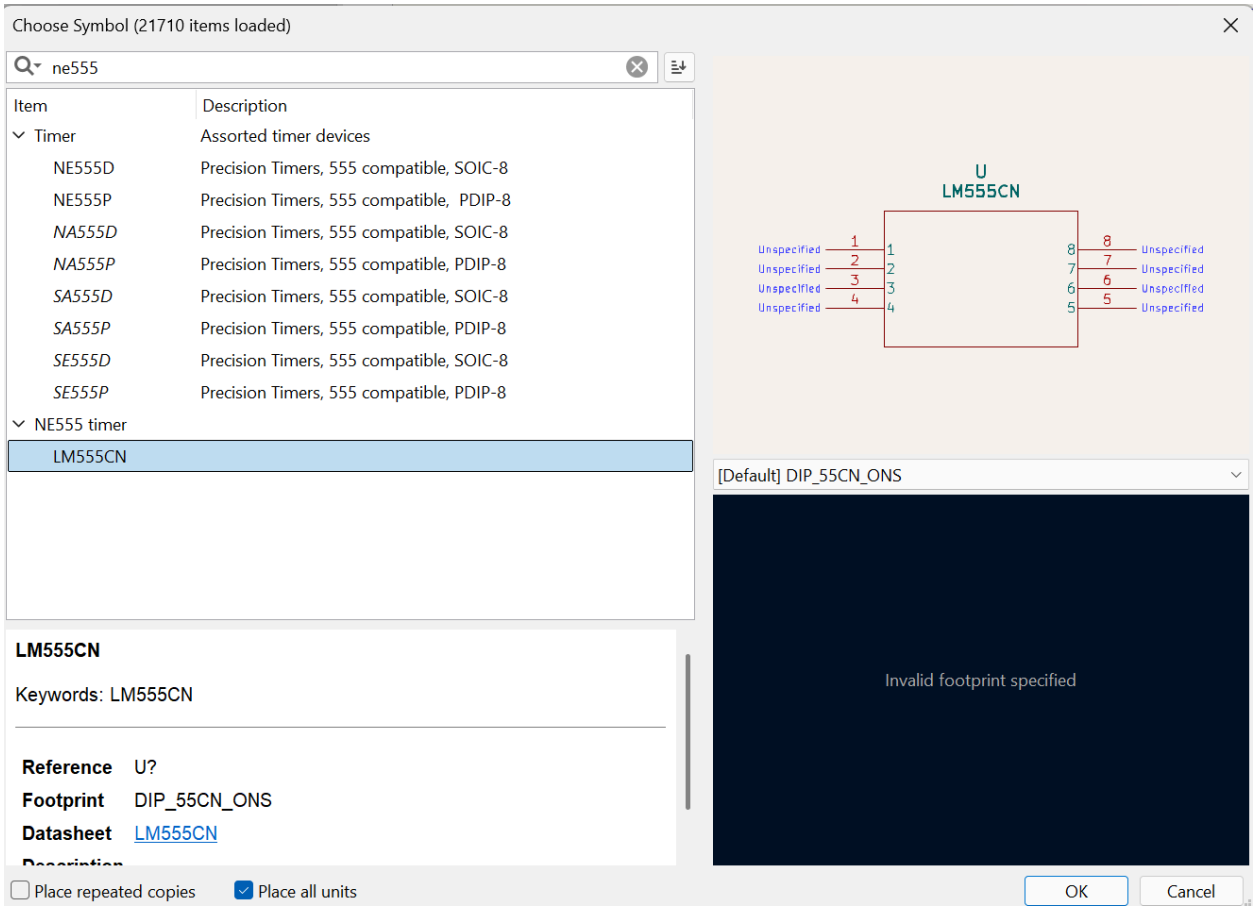
7. Click the folder icon in the bottom left corner
8. Now navigate to the place that you unzipped the EDA/CAD files



9. Select the ".kicad_sym" file and select open



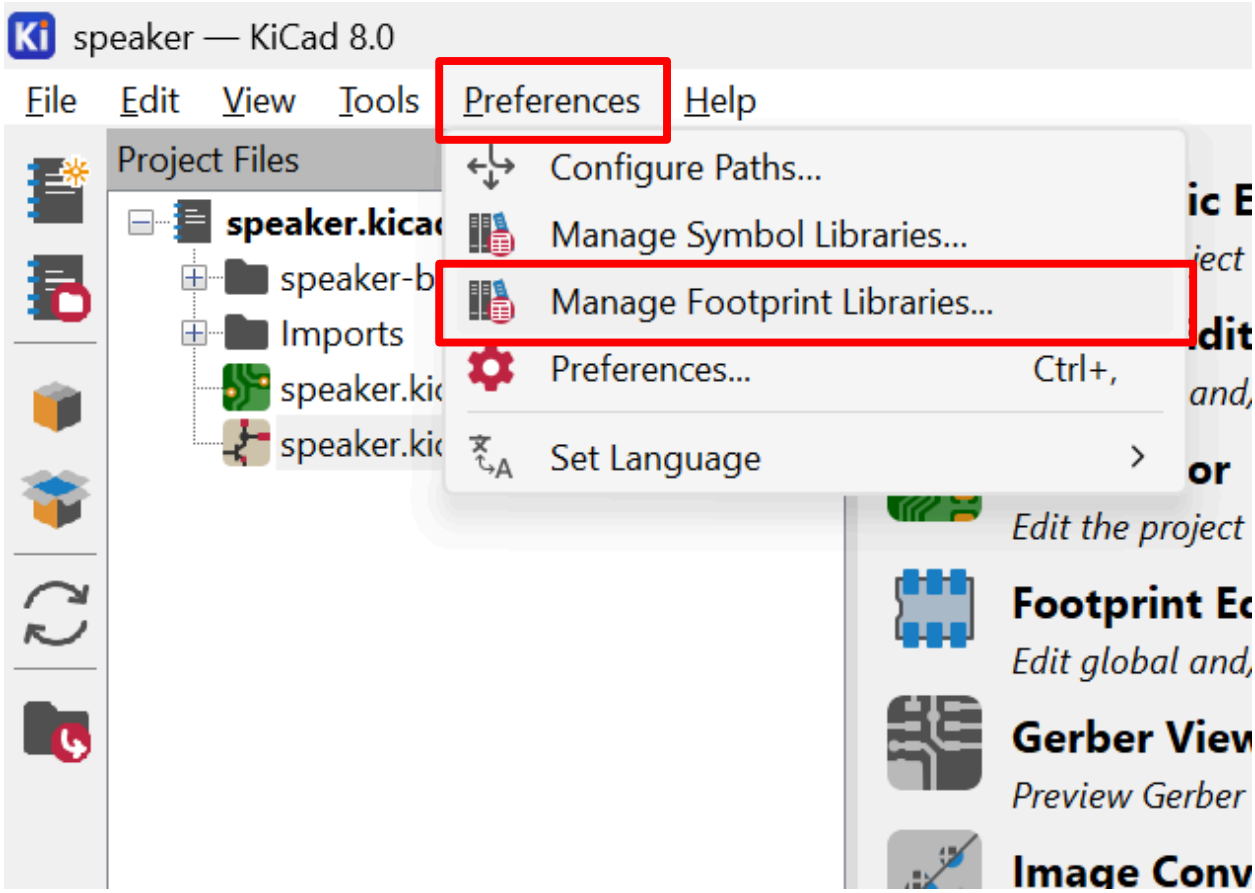
10. Rename the symbol to something relevant (e.g. NE555 timer)



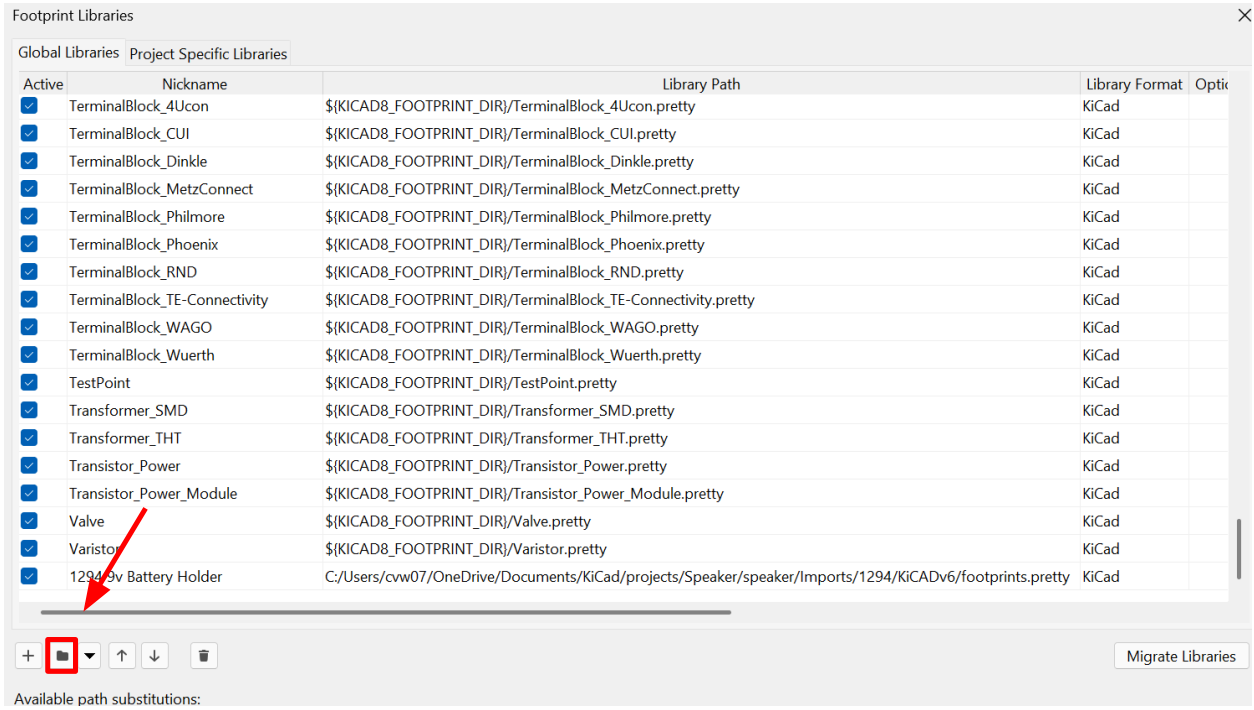
11. Inside KiCad Schematic Editor, add a symbol to your schematic and search for the part you just created

Importing Footprints

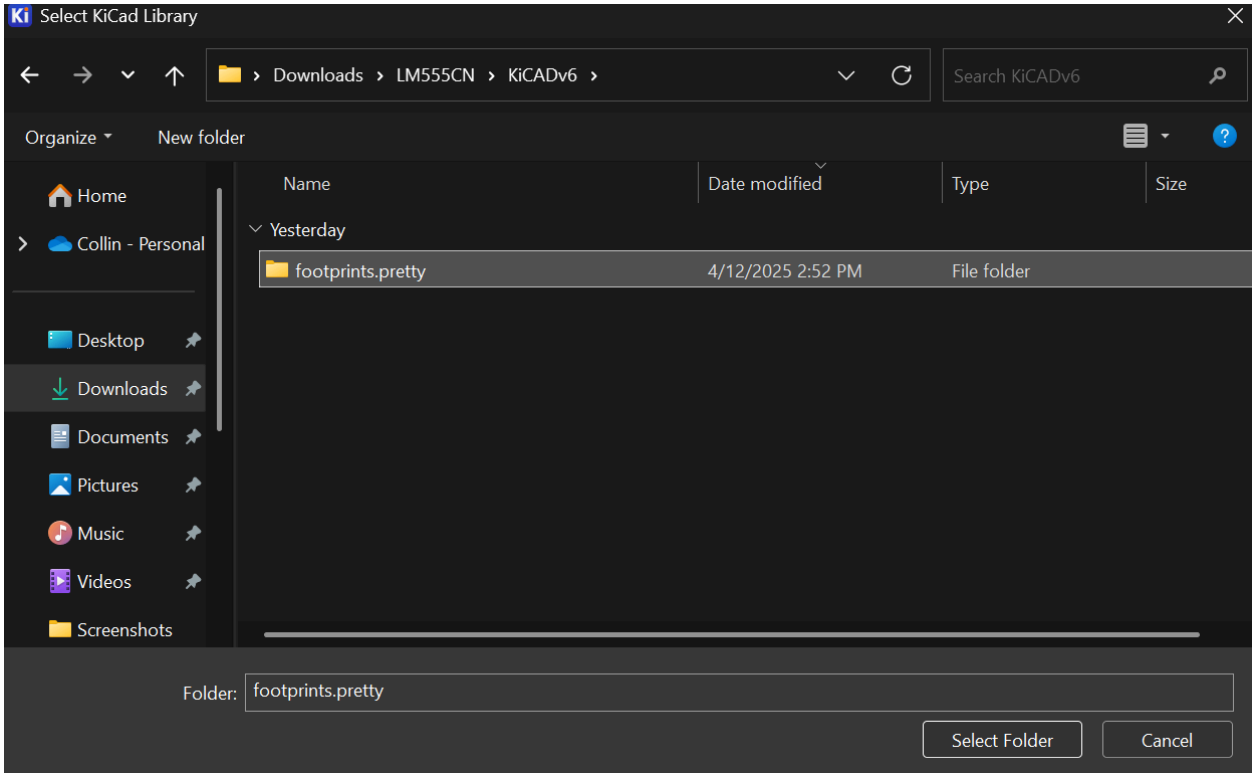
1. Perform steps 1 through 5 of Importing Symbol section



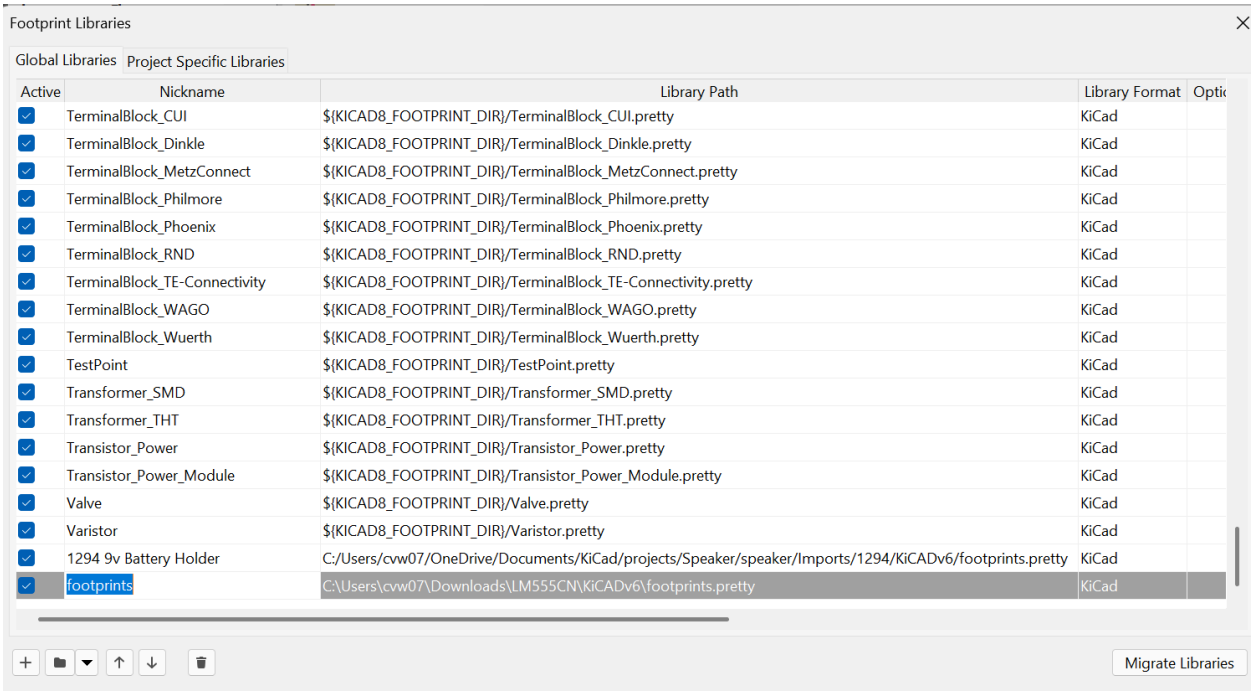
2. To import the Footprint open the current version of KiCad installed on your device and navigate to “Manage Footprint Libraries”



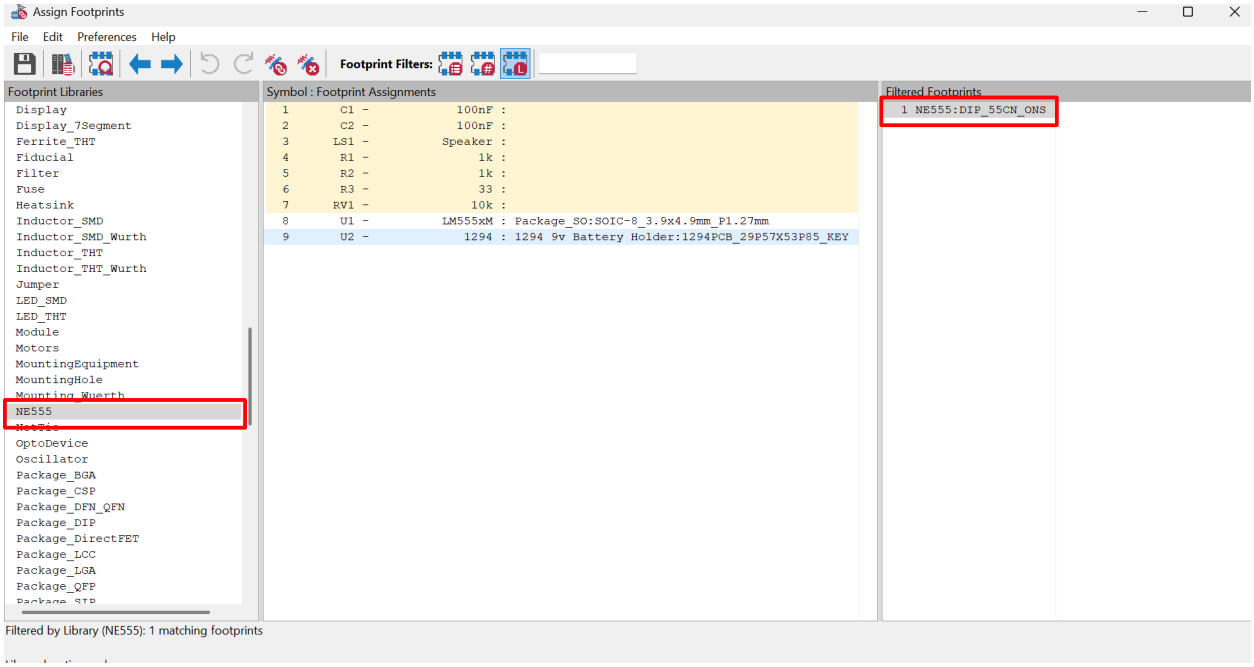
3. Click the folder icon in the bottom left corner
4. Now navigate to the place that you unzipped the EDA/CAD files



5. Navigate to the ".pretty" folder and then Select Folder



6. Rename the symbol to something relevant (e.g. NE555 timer)



7. The footprint should now appear in your footprint library when assigning footprints