To reference properly in GRC, please follow these instructions before placing any components on your schematic to make it so that everybody can open your design files

1. Once creating a project file and schematic, go into Preferences->Manage Symbol Libraries ->Project Specific Libraries and click the add button

Graphical user interface, text, application, email

Description automatically generated

1. Make sure that the active column is checked, the nickname is not important but make it relevant, the library path by default will be wherever your KiCAD libraries are installed by default its:

C:\Users\GRC\Documents\KiCad\6.0\symbols

1. Copy and paste the following text and insert whichever library you want to add

Symbols

${KIPRJMOD}\..\GRCLibrary\Symbols\\*NAME\_OF\_LIBRARY\*.kicad\_sym

Footprints

${KIPRJMOD}\..\GRCLibrary\Footprints\\*NAME\_OF\_LIBRARY\*

Graphical user interface, text, application

Description automatically generated

4)Ensure that you note down the Nickname because that will be our search tool we will pull the files from

5)Click Ok

6)Click CTRL-A and click on the schematic to place a component

7)Search the Nickname and make sure that you ONLY USE THE SCHEMATICS FROM THIS NICKNAME LIBRARY

Graphical user interface, text, application, email

Description automatically generated