

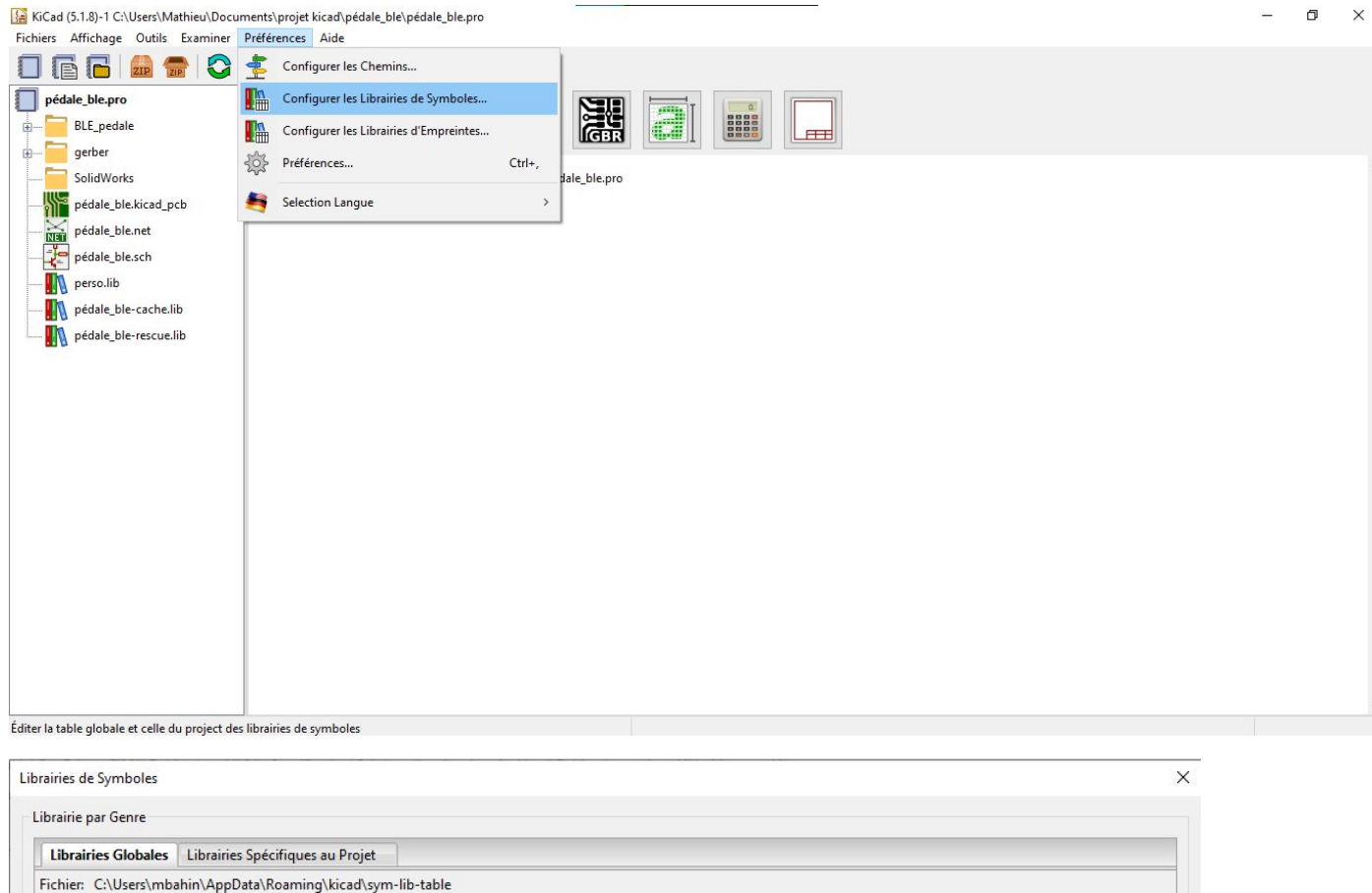
Documentation modification of the Kicad project


From KiCad open the .pro

Load libraries in the project

Modification of the wiring diagram:

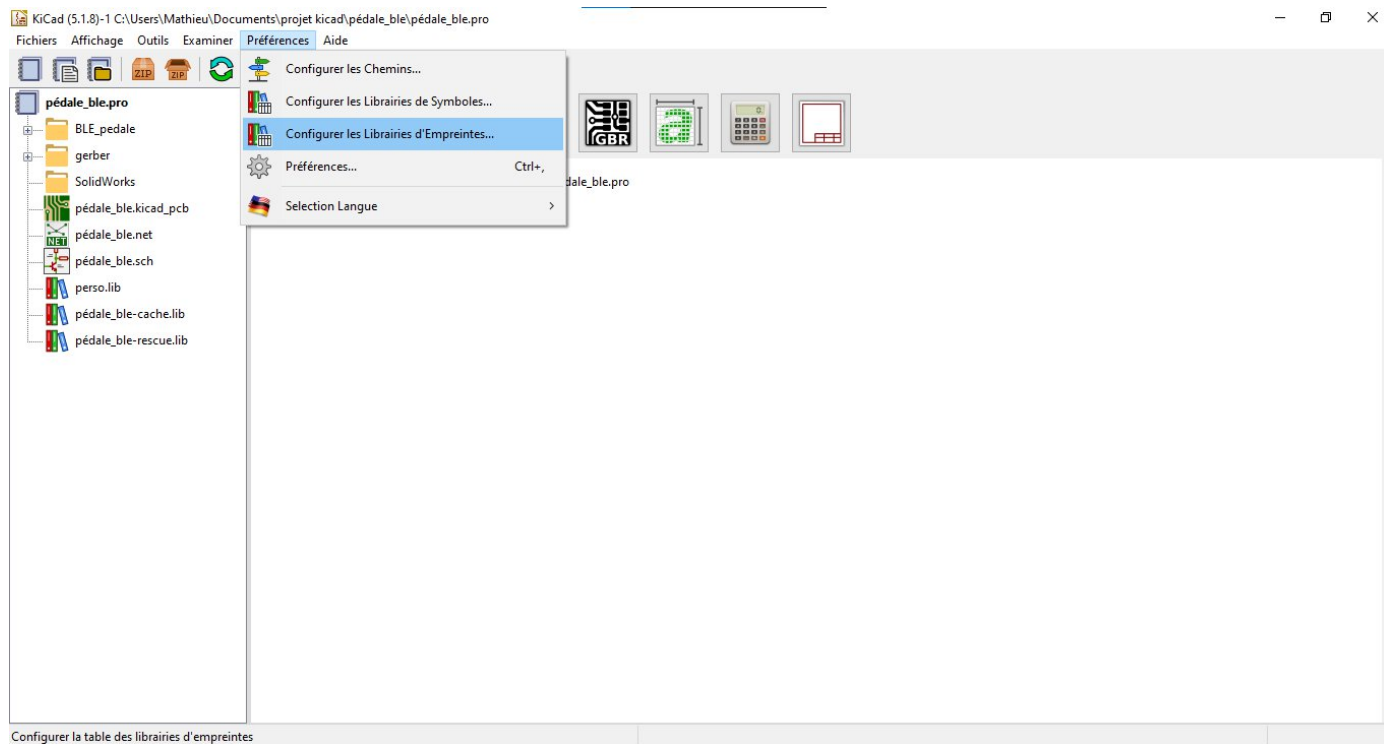
In preferences / Configure symbols path / Global libraries :




- Click on  then select the used libraries
 - o lib_dcm / Ctrl+A Enter

Modification routing :

In preferences / Configure symbols path / Global libraries :



- Click on  then select the used footprints
 - o perso.pretty (selection of folder)

If the libraries are not loaded, some components appear as a box with ? inside (in electronic schematic editing mode).