Gerber and EasyEDA project files

Please note, that the Gerber files have been regenerated and are untested!

If you use the EasyEDA project files you can make changes and generate your own Gerber files.

You can open the project zip files directly in the EasyEDA designer (Std Edition)

      File->Open->EasyEDA

You will get a schematic and a PCB and you can save them to a new project.

The case has been designed for 1.6mm boards with one exception: the label must be a 1mm board.

For this project I recommend ENIG black for the label board and lead-free (RoHS) black for the other boards. Black boards look better behind the acrylic panel.