# PCB Spec

Layers: 2 layers (top and bottom)

Material Details: FR4-Standard

Board Thickness: **0.6mm (not 1.6mm!!)**

Board Dimensions: Width 40.200mm x Height 20.800mm

Surface Finish: Lead Free HASL

Copper Thickness: 1oz (35um)

Soldermask Sides: Top

Soldermask Color: Green

Silkscreen Sides: Both

Silkscreen Color: White

# Description of the files contained:

\*.drl (Drill Data)

\*-drl.rpt (Drill Tool List)

\*-drl\_map.gbr (Drill Map)

\*-Edge.Cuts.gbr (Board Outline)

\*-top.pos (Position of the modules on Top Layer)

**Top Layer**

\*-F.Adhes.gbr (Gerber Top Layer Adhesives)

\*-F.Cu.gbr (Gerber Top Layer)

\*-F.Mask.gbr (Gerber Top Solder Mask)

\*-F.Paste.gbr (Gerber Top Solder Paste for stencil)

\*-F.SilkS.gbr (Gerber Top Silk)

**Bottom Layer**

\*-B.Adhes.gbr (Gerber Bottom Layer Adhesives)

\*-B.Cu.gbr (Gerber Bottom Layer)

\*-B.Mask.gbr (Gerber Bottom Solder Mask)

\*-B.Paste.gbr (Gerber Bottom Solder Paste for stencil)

\*-B.SilkS.gbr (Gerber Bottom Silk)

**BOM file**: fm\_bpf.xlsx

# Note for Assembly

All the components are **chosen carefully** including passives. So **use the part as-is** where BOM column “Compatible parts can be used?” is marked “N”. If you cannot procure the parts as specified in BOM file, please let me know the suggested parts and their datasheets.

All the parts are mount on the TOP SIDE.

Because the PCB is thin, soldering on bottom side for the SMA connector is not necessary (top side **is** necessary).