# PCB Spec

Layers: 4 layers

Material Details: FR4-Standard

Board Thickness: 1.6mm

Board Dimensions: Width 69.600mm x Height 43.800mm

Surface Finish: Lead Free HASL

Copper Thickness: 1oz (35um)

Soldermask Sides: Both

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 (Top Layer Module Positions)

**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)

**2nd layer**

\*-In1.Cu.gbr (Gerber 2nd layer)

**3rd layer**

\*-In2.Cu.gbr (Gerber 3rd layer)

**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:** zturn\_adc.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.

Do not load R29, R30 (and NO SOLDER).

4 center pads of U1, U6 should also be soldered.

CN3 is towards OUTSIDE of the PCB.

Take care of the orientation when mounting the following parts: U1, U2, U3, U4, U5, U6, X1, T1. Pin #1 is marked with triangle “△”.