# 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)

1st 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)

4th 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 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 “△”.