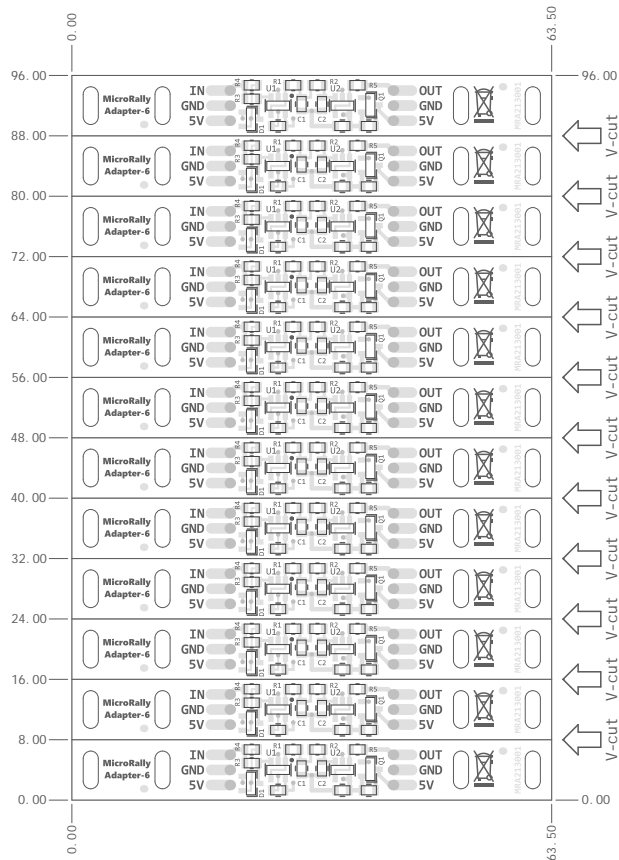
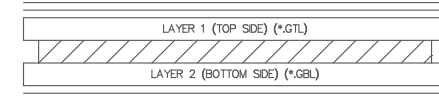


# PANEL GERBER NOTES



1.0 OZ L1  
1.50 mm Core - FR4  
1.0 OZ L2



PASTE (TOP SIDE) (\*GTP)  
SILKSCREEN (TOP SIDE) (\*GTO)  
SOLDERMASK (TOP SIDE) (\*GTS)  
1.6 mm +/- 10%  
LAYER DETAIL  
2 LAYERS  
SOLDERMASK (BOTTOM SIDE) (\*GBS)

## NOTES: (UNLESS OTHERWISE SPECIFIED)

- THIS IS 2 LAYER BOARD
- MATERIAL: FR4, TG 150 DEGREE C MIN
- FR4 DIELECTRIC CONSTANT NOT SPECIFIED
- FINISHED BOARD THICKNESS TO BE 1.60MM +/- 10%
- TRACE WIDTHS IN ARTWORK ARE FINISHED SIZES
- SEE FILM FOR LAYER SEQUENCE AND COPPER THICKNESSES (SHOWN BEFORE PLATING)
- MIN TRACE/SPACE 0.25/0.30MM
- SEE DRILL CHART FOR FINISHED HOLE SIZES
- MIN DRILL 0.30MM
- HOLE TOLERANCE IS +/-3MIL UNLESS OTHERWISE SPECIFIED  
HOLE COPPER THICKNESS MIN 0.7MIL  
SLOT TOLERANCE +/-0.1MM  
BORDER OUTLINE TOLERANCE +/-0.15MM
- SURFACE PLATING: HASL, Pb FREE
- SOLDERMASK: LPI, BOTH SIDES. COLOR GREEN
- SILKSCREEN: TOP AND SIDE. COLOR WHITE
- ALL BOARDS MUST BE ELECTRICALLY TESTED FOR ISOLATION (SHORTS) AND CONTINUITY (OPENS)

|                                    |             |
|------------------------------------|-------------|
| Project: MR Adapter-6 Panel        |             |
| Author: Andis Jargans              | Revision: 1 |
| Date: 28.07.2021                   | Size: A4    |
| File: MR_Adapter-6_r1_PANEL.PcbDoc | MicroRally  |