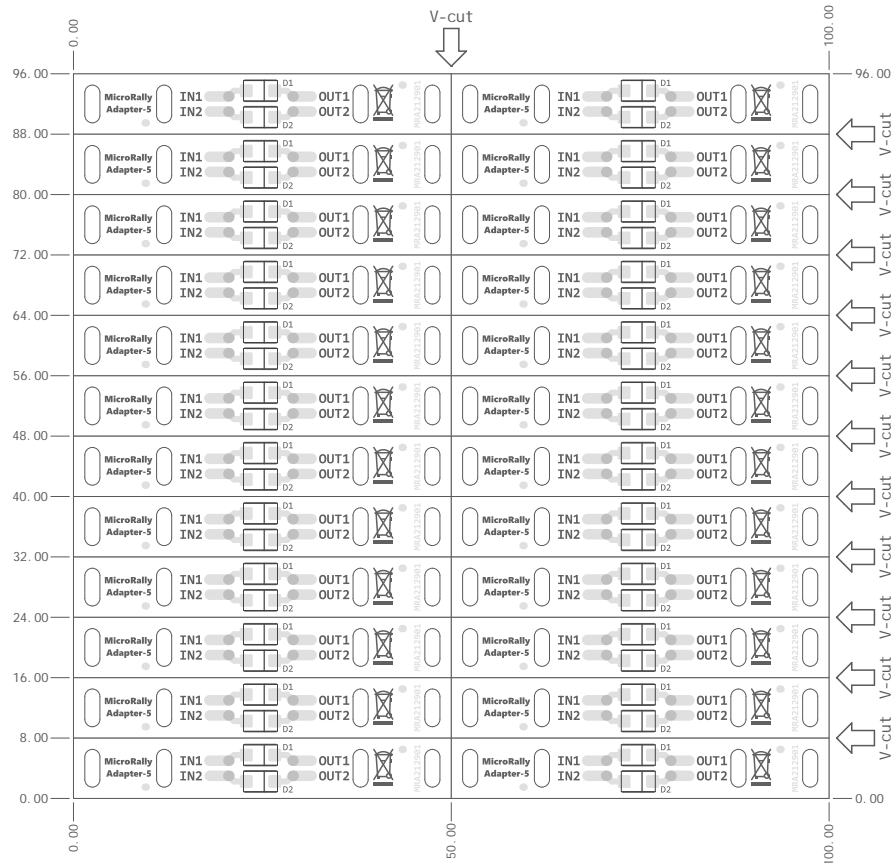
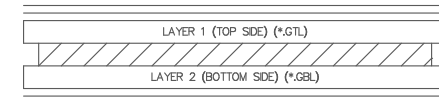


PANEL GERBER NOTES



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)
SILKSCREEN (BOTTOM SIDE) (*GB0)
PASTE (BOTTOM SIDE) (*GBP)

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 0.80/0.60MM
- SEE DRILL CHART FOR FINISHED HOLE SIZES
- MIN DRILL 0.80MM
- 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 BOTTOM SIDE. COLOR WHITE
- ALL BOARDS MUST BE ELECTRICALLY TESTED FOR ISOLATION (SHORTS) AND CONTINUITY (OPENS)

Project: MR Adapter-5 Panel	
Author: Andis Jargans	Revision: 1
Date: 28.07.2021	Size: A4
File: MR_Adapter-5_r1_PANEL.PcbDoc	MicroRally