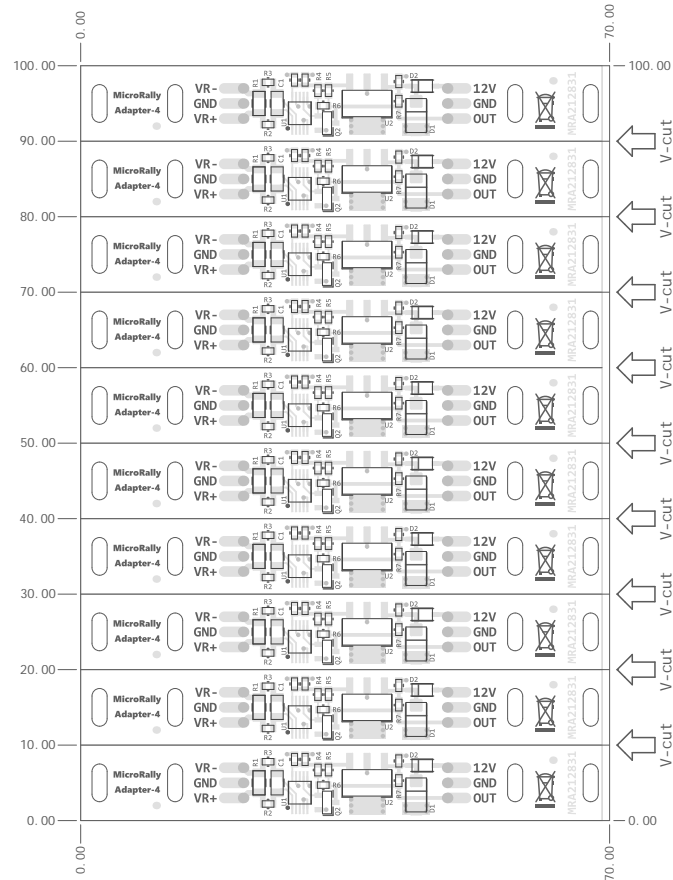
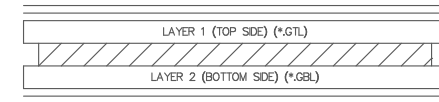


PANEL GERBER NOTES



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 0.25/0.20MM
- SEE DRILL CHART FOR FINISHED HOLE SIZES
- MIN DRILL 0.30MM
- HOLE TOLERANCE IS +/-0.3MIL 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 SIDE. COLOR WHITE
- ALL BOARDS MUST BE ELECTRICALLY TESTED FOR ISOLATION (SHORTS) AND CONTINUITY (OPENS)

Project: MR Adapter-4 Panel	
Author: Andis Jargans	Revision: 1
Date: 29.07.2021	Size: A4
File: MR_Adapter-4_r1_PANEL.PcbDoc	MicroRally