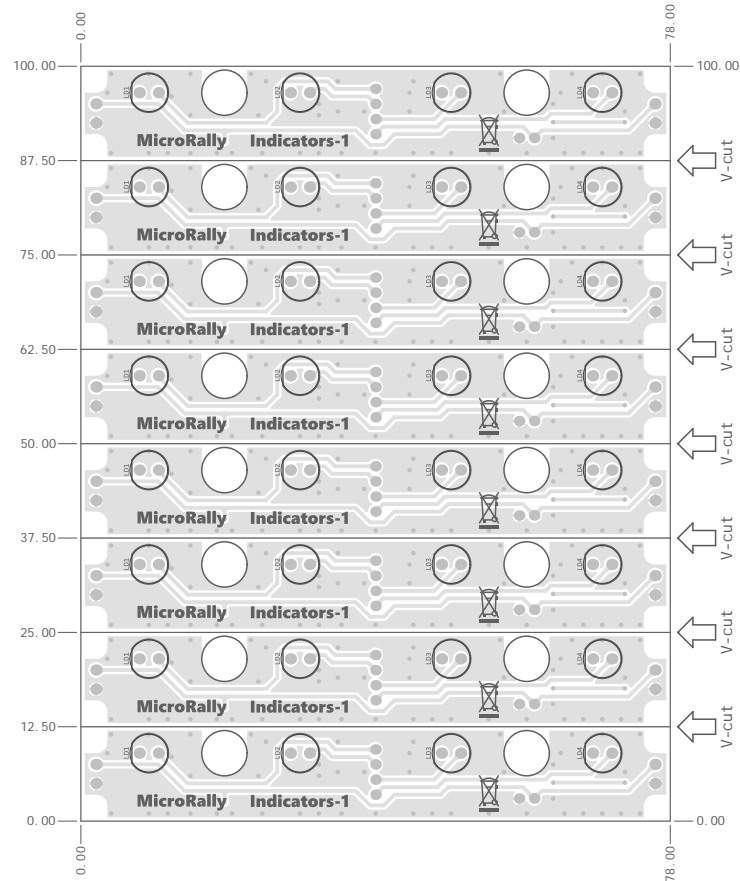
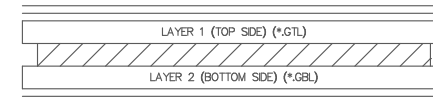


# PANEL GERBER NOTES



1.0 OZ L1  
1.50 mm Core – FR4  
1.0 OZ L2



1.6 mm  $\pm$  10%  
LAYER DETAIL  
2 LAYERS  
SILKSCREEN (TOP SIDE) (\*GTO)  
SOLDERMASK (TOP SIDE) (\*GTS)  
SOLDERMASK (BOTTOM SIDE) (\*GBS)  
SILKSCREEN (BOTTOM SIDE) (\*GBO)  
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  $\pm$  10%
- TRACE WIDTHS IN ARTWORK ARE FINISHED SIZES
- SEE FILM FOR LAYER SEQUENCE AND COPPER THICKNESSES (SHOWN BEFORE PLATING)
- MIN TRACE/SPACE 0.30/0.20MM
- SEE DRILL CHART FOR FINISHED HOLE SIZES
- MIN DRILL 0.30MM
- HOLE TOLERANCE IS  $\pm$  0.03MM UNLESS OTHERWISE SPECIFIED  
HOLE COPPER THICKNESS MIN 0.7MIL  
SLOT TOLERANCE  $\pm$  0.1MM  
BORDER OUTLINE TOLERANCE  $\pm$  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 Indicators-1 Panel	
Author: Andis Jargans	Revision: 1
Date: 24.09.2021	Size: A4
File: MR_Indicators-1_r1_PANEL.PcbDoc	MicroRally