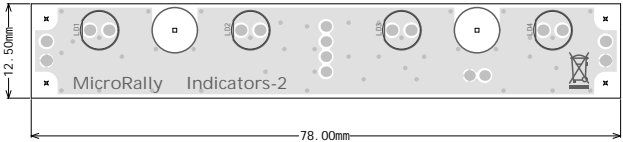
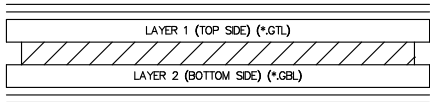


GERBER NOTES

Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad
▽	42	0.300mm	PTH	Round	Via
✱	10	0.800mm	PTH	Round	Pad
○	8	0.900mm	PTH	Round	Pad
✱	4	1.152mm	NPTH	Round	Pad
□	2	3.200mm	NPTH	Round	Pad
	66 Total				



1.0 OZ L1
1.50 mm Core – FR4
1.0 OZ L2



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

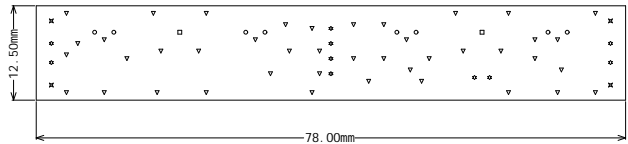
NOTES: (UNLESS OTHERWISE SPECIFIED)

- 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 0.30/0.20MM
- 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 BOTTOM SIDE. COLOR WHITE
- ALL BOARDS MUST BE ELECTRICALLY TESTED FOR ISOLATION (SHORTS) AND CONTINUITY (OPENS)

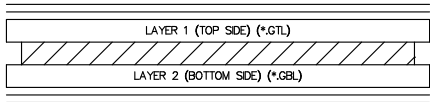
Project: MR Indicators-2	
Author: Andis Jargans	Revision: 1
Date: 25.09.2021	Size: A4
File: MR_Indicators-2_r1.PcbDoc	MicroRally

GERBER NOTES

Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad
▽	42	0.300mm	PTH	Round	Via
✱	10	0.800mm	PTH	Round	Pad
○	8	0.900mm	PTH	Round	Pad
⊠	4	1.152mm	NPTH	Round	Pad
□	2	3.200mm	NPTH	Round	Pad
	66 Total				



1.0 OZ L1
1.50 mm Core – FR4
1.0 OZ L2



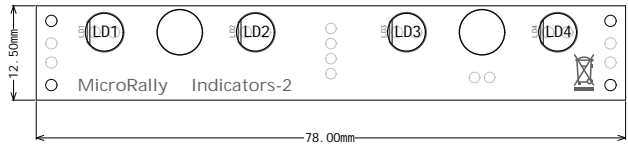
1.6 mm +/- 10%
SILKSCREEN (TOP SIDE) (*GTO)
SOLDERMASK (TOP SIDE) (*GTS)
LAYER DETAIL
2 LAYERS
SOLDERMASK (BOTTOM SIDE) (*GBS)
SILKSCREEN (BOTTOM SIDE) (*GBO)
PASTE (BOTTOM SIDE) (*GBP)

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 +/- 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 +/-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 Indicators-2	
Author: Andis Jargans	Revision: 1
Date: 25.09.2021	Size: A4
File: MR_Indicators-2_r1.PcbDoc	MicroRally

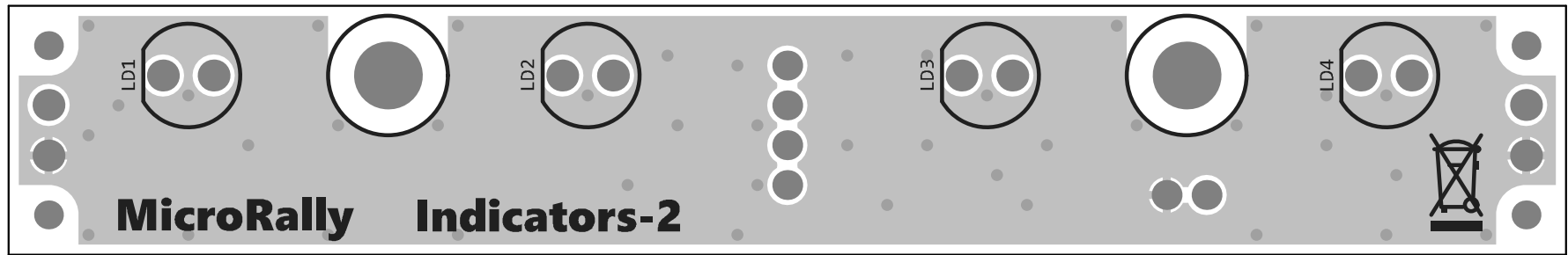
ASSEMBLY NOTES

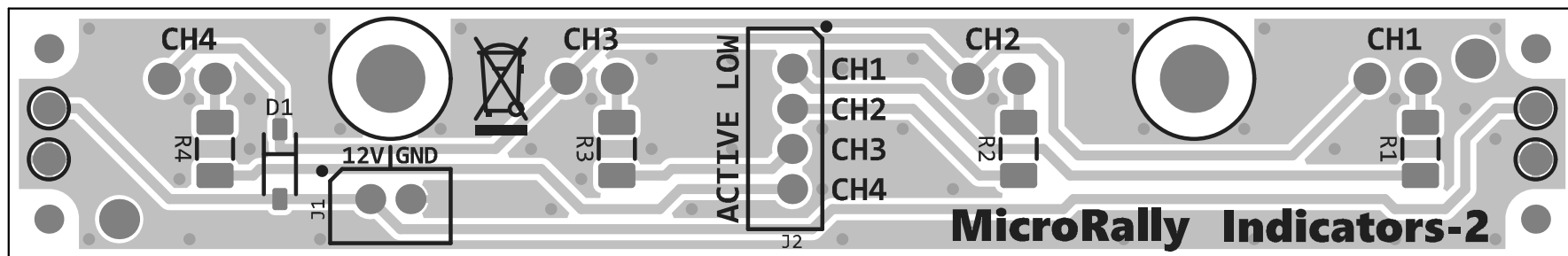


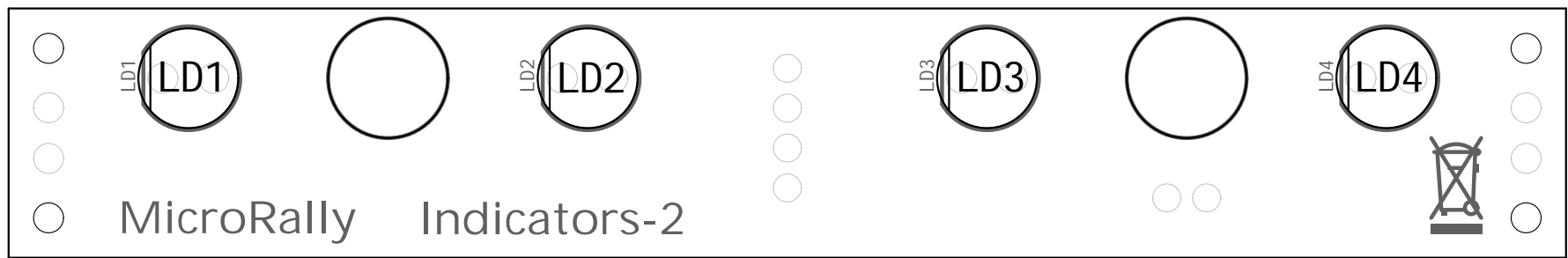
DAISY CHAIN BPARDS
USE 2.54MM PITCH PADS TO DAISY CHAIN ADJECENT BOARD POWER
IN THIS WAY ONLY ONE 2 PIN CONNECTOR IS NECCESERY FOR COMMON GROUND AND POWER INPUT

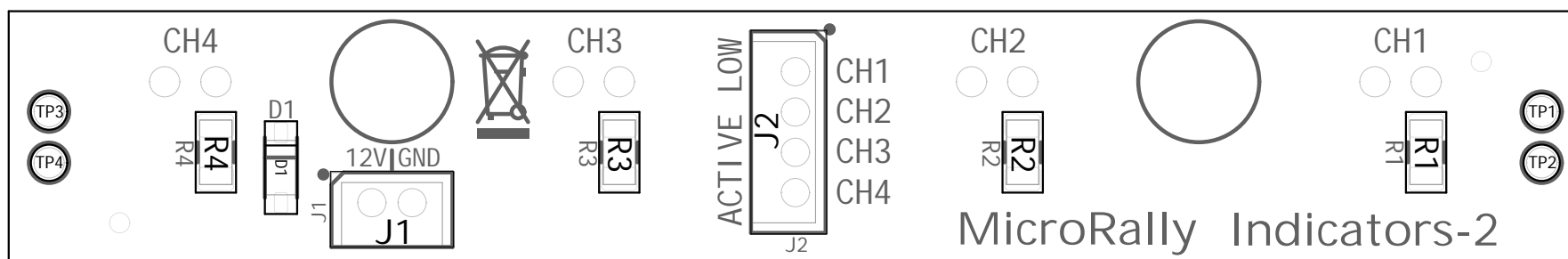
LED BRI GHTNESS
LED BRI GHTNESS CAN BE CHANGED BY CHANGEING CURRENT LIMITING RESI STOR.

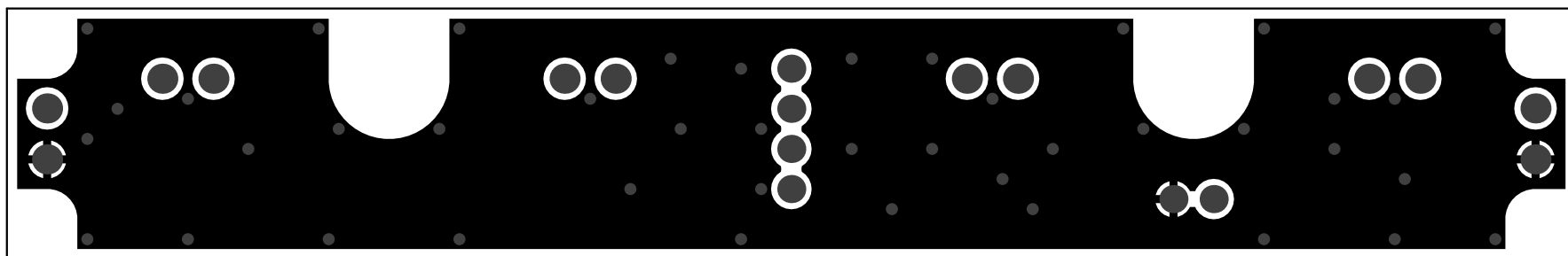
Project: MR Indi cators-2	
Author: Andi s Jargans	Revi si on: 1
Date: 25. 09. 2021	Si ze: A4
Fi le: MR_Indi cators-2_r1. PcbDoc	Mi croRal ly

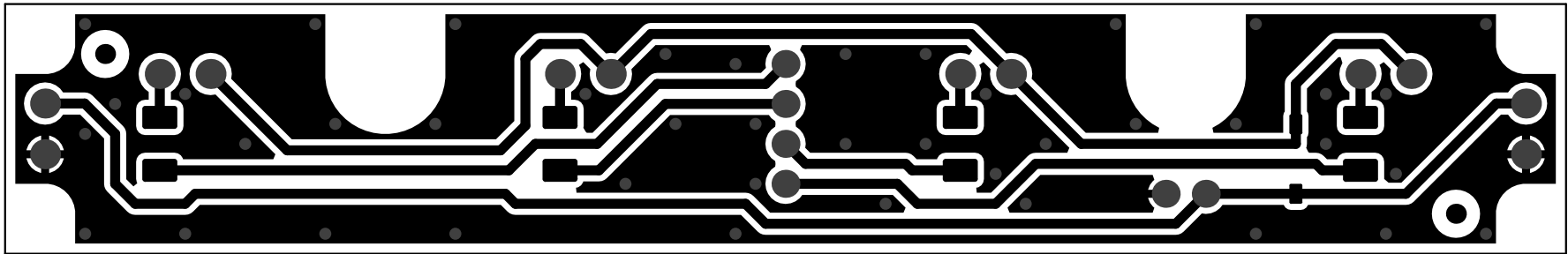


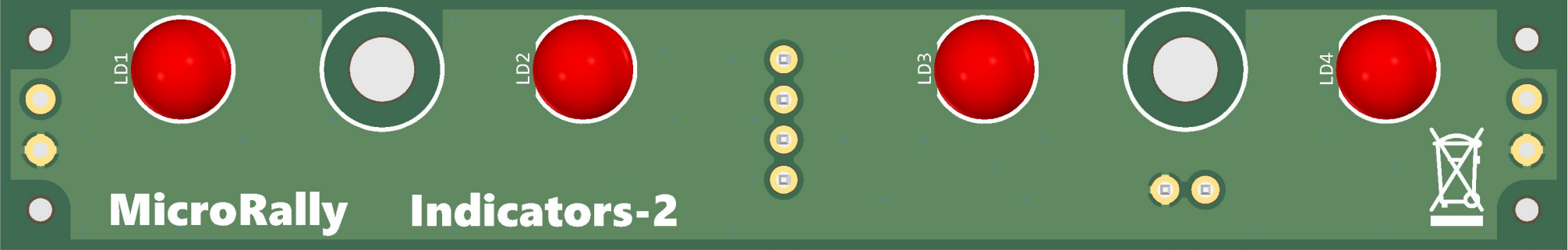


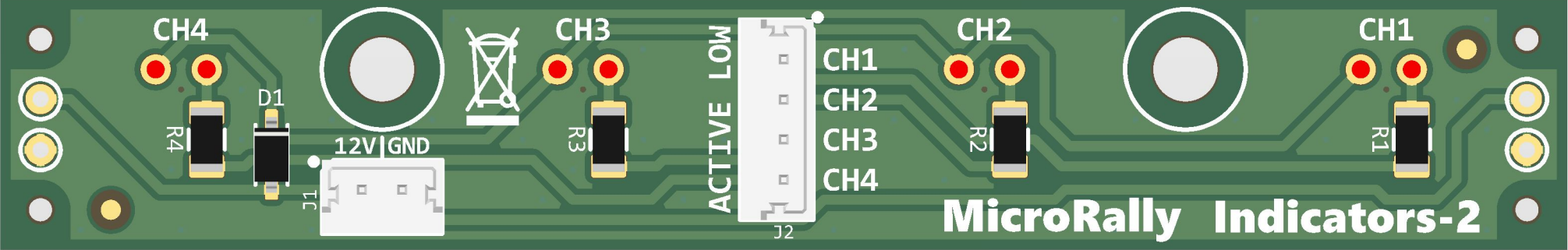


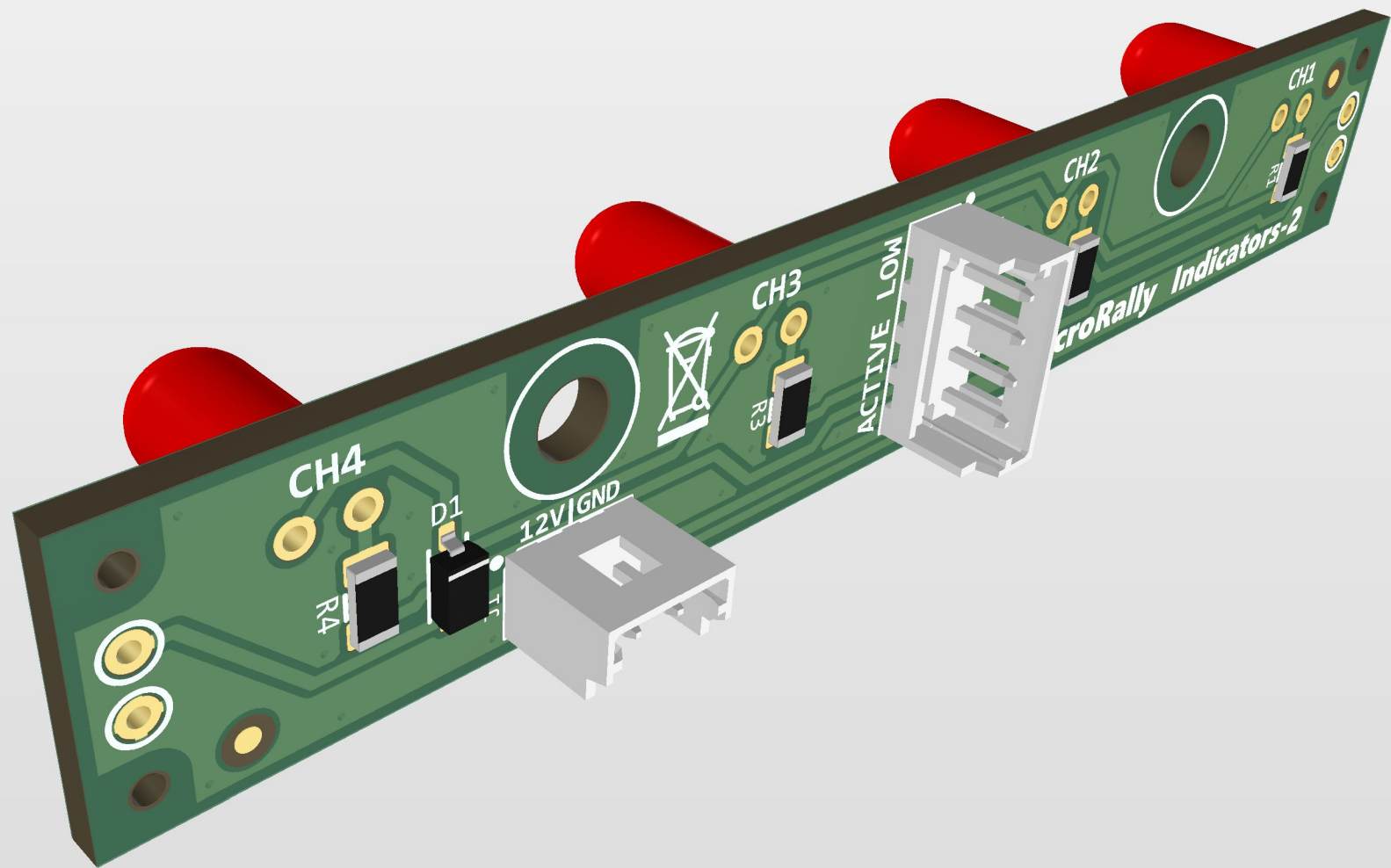


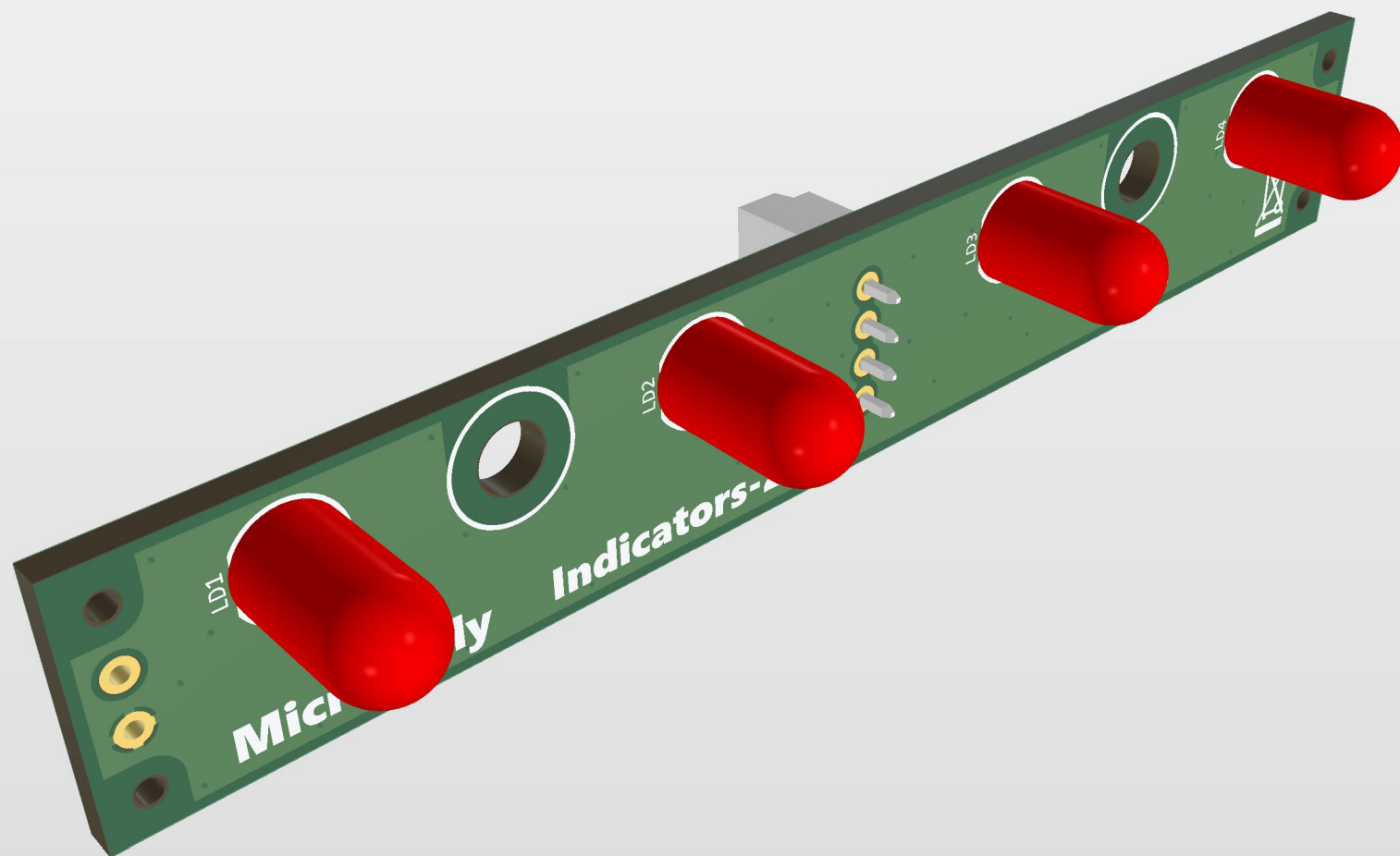




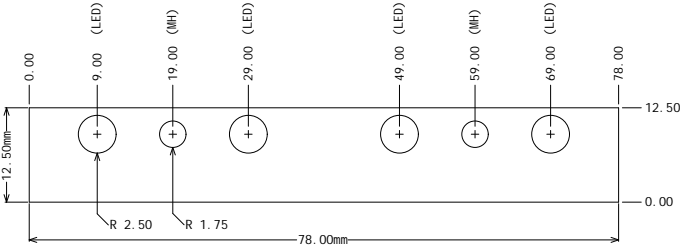








PANEL CUTOUT NOTES



Project: MR Indicators-2	
Author: Andi s Jargans	Revi si on: 1
Date: 25. 09. 2021	Si ze: A4
Fi le: MR_Indi cators-2_r1. PcbDoc	Mi croRal ly