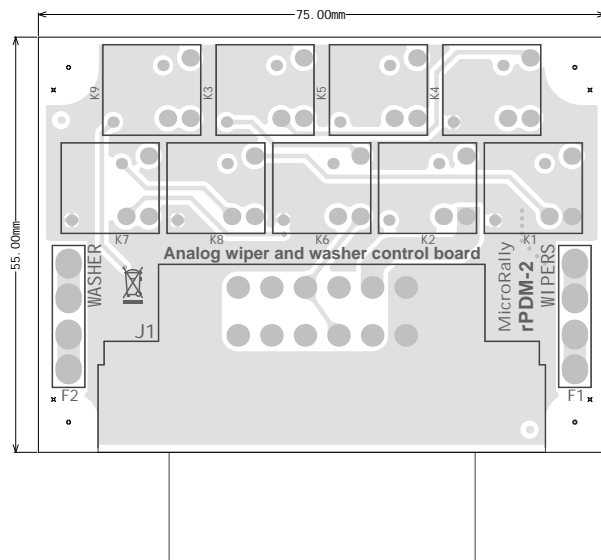
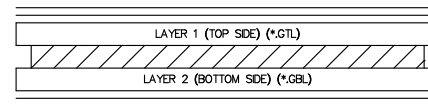


GERBER NOTES



Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad
⊗	10	0.300mm	PTH	Round	Via
○	18	0.800mm	PTH	Round	Pad
⊗	4	1.152mm	NPTH	Round	Pad
□	27	1.300mm	PTH	Round	Pad
⊗	12	1.700mm	PTH	Round	Pad
▽	8	2.500mm	PTH	Round	Pad
⊕	4	3.200mm	NPTH	Round	Pad
	83 Total				

1.0 OZ L1
1.50 mm Core - FR4
1.0 OZ L2



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)

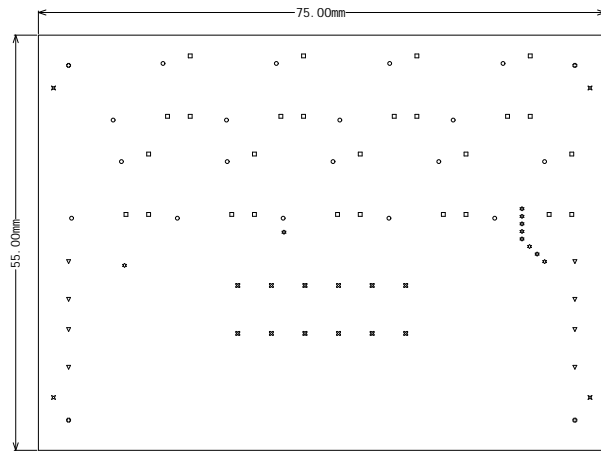
NOTES: (UNLESS OTHERWISE SPECIFIED)

- 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 0.60/0.30MM
- 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 BOTTOM SIDE. COLOR WHITE
- ALL BOARDS MUST BE ELECTRICALLY TESTED FOR ISOLATION (SHORTS) AND CONTINUITY (OPENS)

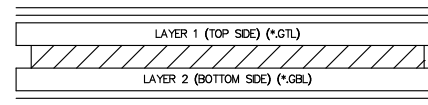
Project: rPDM-2	
Author: Andis Jargans	Revision: 1
Date: 08.04.2021	Size: A4
File: rPDM-2_r1.PcbDoc	MicroRally

GERBER NOTES

Symbol	Count	Hole Size	Plated	Hole Type	Via/Pad
⊗	10	0.300mm	PTH	Round	Via
○	18	0.800mm	PTH	Round	Pad
⊗	4	1.152mm	NPTH	Round	Pad
□	27	1.300mm	PTH	Round	Pad
⊗	12	1.700mm	PTH	Round	Pad
▽	8	2.500mm	PTH	Round	Pad
⊕	4	3.200mm	NPTH	Round	Pad
	83 Total				



1.0 OZ L1
 1.50 mm Core - FR4
 1.0 OZ L2



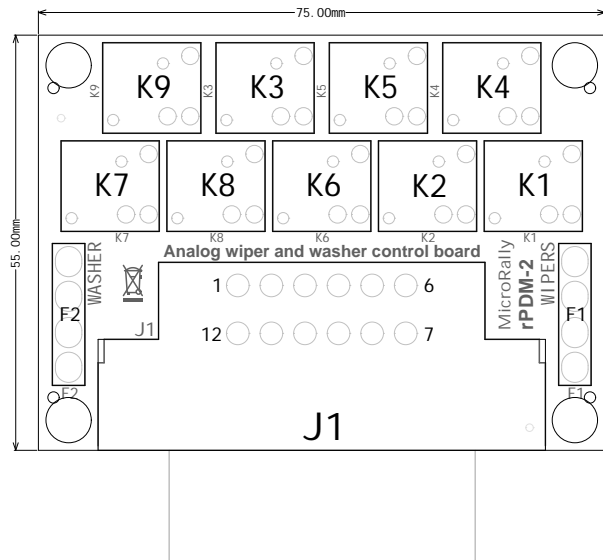
↑ SILKSCREEN (TOP SIDE) (*GTO)
 SOLDERMASK (TOP SIDE) (*GTS)
 1.6 mm +/- 10% LAYER DETAIL
 2 LAYERS
 ↓ SOLDERMASK (BOTTOM SIDE) (*GBS)
 SILKSCREEN (BOTTOM SIDE) (*GBO)
 PASTE (BOTTOM SIDE) (*GBP)

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 0.60/0.30MM
- 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 BOTTOM SIDE. COLOR WHITE
- ALL BOARDS MUST BE ELECTRICALLY TESTED FOR ISOLATION (SHORTS) AND CONTINUITY (OPENS)

Project: rPDM-2	
Author: Andis Jargans	Revision: 1
Date: 08.04.2021	Size: A4
File: rPDM-2_r1.PcbDoc	Microwally

ASSEMBLY NOTES



Project: rPDM-2	
Author: Andi s Jargans	Revi si on: 1
Date: 08.04.2021	Si ze: A4
Fi le: rPDM-2_r1.PcbDoc	Mi croRal ly

