

REVISIONS				
ZONE	REV.	DESCRIPTION	NAME/DATE	APPD/DATE

Layer	Name	Material	Thickness	Constant	Board Layer Stack	Board Layer Stack	Board Layer Stack
	Top Overlay						
	Top Solder	Solder Resist	0.50mil	3.5			
1	Top Layer		1.38mil				
	Dielectric1	PP-022	8.20mil	4.5			
2	Inner Layer 1		1.38mil				
	Dielectric4	Core-042	39.00mil	4.6			
3	Inner Layer 2		1.38mil				
	Dielectric7	PP-022	8.20mil	4.5			
4	Bottom Layer		1.38mil				
	Bottom Solder	Solder Resist	0.50mil	3.5			
	Bottom Overlay						

Total board thickness (Board Layer Stack): 61.91mil

1. RoHS COMPLIANCE: YES ☒ NO ☐
ALL MATERIALS, LAMINATES, RESINS, METALIZATIONS, INKS AND SOLDERMASK TO BE COMPLIANT TO EU RoHS DIRECTIVE 2002/95/EC. LAMINATE AND RESIN MATERIALS TO HAVE A Tg >170°C, Td >350°C AND WHEN CONSTRUCTED, BE ABLE TO WITHSTAND 6X THERMAL CYCLES AT 260°C. VENDOR TO ADD MARKING Pb-FREE OR USE APPROVED SYMBOL PER IPC-1066 TO PRIMARY SIDE SILKSCREEN. MARKING SHALL BE ADDED TO PRIMARY SIDE ETCH ONLY IF SILKSCREEN IS NOT REQUIRED.

2. IPC SPECIFICATION:
MANUFACTURE BOARD IN ACCORDANCE WITH PERFORMANCE STANDARD IPC-6011/6012 CLASS 2. BOARD TO BE INSPECTED PER IPC-600-A CLASS 2 LATEST REV.

3. MATERIAL SPECIFICATIONS
GLASS FIBER EPOXY LAMINATE AND PREPREG MATERIALS PER IPC-4101 (CURRENT REVISION). ALL MATERIALS MUST MEET RoHS COMPLIANCE CRITERIA. SEE NOTE RoHS COMPLIANCE

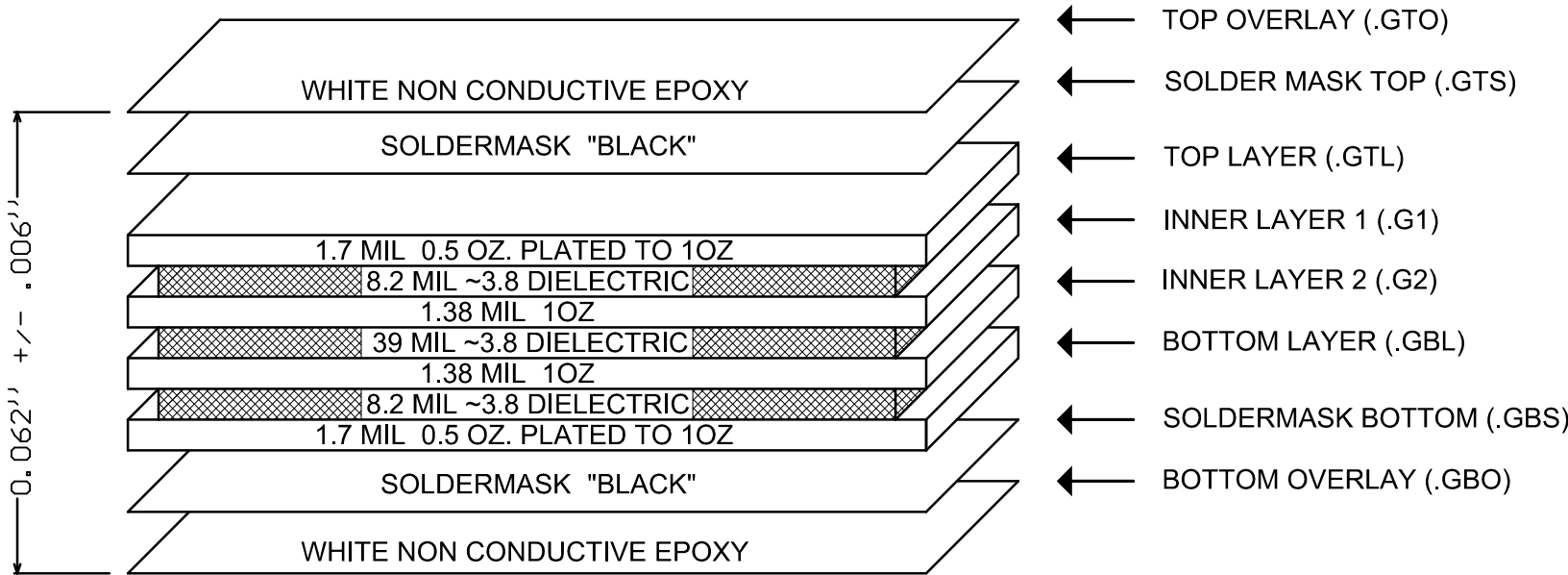
4. PCB THICKNESS:
CARD PROFILE: FINISHED. CARD THICKNESS : 0.062" +/- .006".

5. VIA, HOLE SIZE, FINISH AND THICKNESS:
PLATED HOLE SIZES SHOWN IN LEGEND ARE FINISHED SIZES +/- 0.002", AFTER PLATING. NON-PLATED HOLE SIZES SHOWN IN LEGEND ARE +/-0.002"

6. COPPER WEIGHT (OZ) PER LAYER:
COPPER PLATING -OUTER LAYERS SHALL HAVE A MINIMUM OF 1/2 OZ. COPPER CLADDING AND SHALL BE A MINIMUM OF 1 OZ. AFTER PLATING. FINISHED HOLES SHALL HAVE AN AVERAGE COPPER THICKNESS OF 0.001 AND HAVE A MINIMUM THICKNESS OF 0.0008. WHEN USED, INNER LAYERS SHALL BE A MINIMUM OF 1 OZ

7. SURFACE FINISH:
COPPER FINISH SHALL BE ELECTROLESS NICKEL/IMMERSION GOLD (ENIG) (100-300 MICRO-INCHES) / IMMERSION GOLD FLASH (2-10 MICROINCHES)

8. LAYER STACK
LAYER COUNT: 2 ☐ 4 ☒ 6 ☐
TECHNOLOGY LINE/SPACE : 6 MIL LINE/ 6 MIL SPACE (± 20%)
SMT USED : TOP ☒ BOTTOM ☒
ELECTRICAL TEST : REQUIRED, 100% FOR SHORTS AND OPENS
LAYER STACKUP BELOW:



9. SOLDER MASK SPECIFICATION:
SOLDERMASK TOP AND BOTTOM SIDE OF CARD USING LIQUID PHOTO IMAGEABLE MASK MATERIAL OVER BARE COPPER. PER IPC-SM-840. MASK ARTWORKS PROVIDED ARE 1:1 WITH PAD SIZES. VENDOR SHALL MODIFY TO OBTAIN MINIMUM PAD TO MASK CLEARANCE. SOLDERMASK COLOR "BLACK" (Amendment: to Allow for ENIG before Solder Mask)

10. SILKSCREEN:
SILKSCREEN CARD PER ARTWORK PROVIDED USING NON-CONDUCTIVE WHITE EPOXY INK. VENDOR TO MODIFY SILKSCREEN ARTWORK (CLIP), SUCH THAT NO SILKSCREEN APPEARS ON EXPOSED METALLIC AREAS.

11. UL REQUIREMENTS
BOARDS SHALL BE PURCHASED FROM UL RECOGNIZED VENDORS ONLY AND SHALL BE MARKED IN COPPER ON SECONDARY SIDE OF CARD WITH VENDORS UL IDENTITY, FLAMMABILITY RATING (94-V0), AND DATE CODE (WWYY).

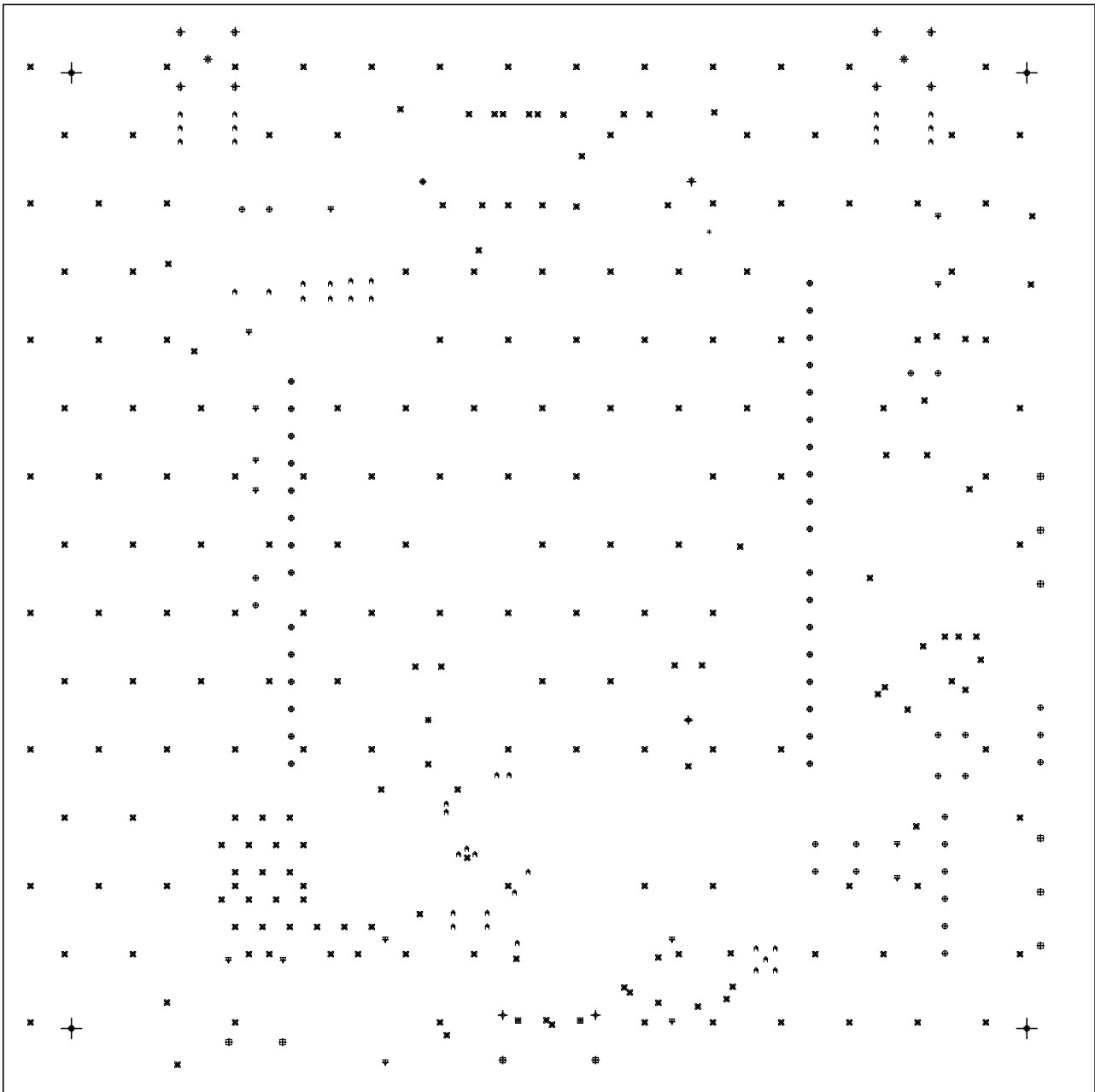
12. ARRAYS DIMENSIONS (PANELIZATION):
IF DIMENSIONS ARE NOT 100% EXPLICIT PLEASE USE GERBER BOARD OUTLINE CONTRACT MANUFACTURE ALLOWED TO PANALIZE PER MACHINE REQUIREMENTS.

13. CONTROLLED IMPEDANCE REQUIREMENTS: YES ☐ NO ☒
NET NAME:
IMPEDANCE REQUIRED:
TOLERANCE:

14. REMOVE BURRS AND SHARP EDGES

Symbol	Count	Hole Size	Hole Length	Routed Path Length	Plated	Hole Type
⌘	1	41.34mil (1.050mm)	-	-	NPTH	Round
⊙	1	44.00mil (1.118mm)	-	-	NPTH	Round
⊛	1	61.02mil (1.550mm)	-	-	NPTH	Round
▽	1	64.00mil (1.626mm)	-	-	NPTH	Round
⊞	2	25.59mil (0.650mm)	-	-	NPTH	Round
⊞	2	55.12mil (1.400mm)	23.62mil (0.600mm)	-31.50mil (-0.800mm)	PTH	Slot
⌘	2	59.06mil (1.500mm)	-	-	PTH	Round
◇	2	66.93mil (1.700mm)	23.62mil (0.600mm)	-43.31mil (-1.100mm)	PTH	Slot
⊙	4	150.00mil (3.810mm)	-	-	NPTH	Round
□	8	51.18mil (1.300mm)	-	-	PTH	Round
B	8	62.99mil (1.600mm)	-	-	PTH	Round
▽	16	40.00mil (1.016mm)	-	-	PTH	Round
○	23	40.16mil (1.020mm)	-	-	PTH	Round
⊙	32	39.37mil (1.000mm)	-	-	PTH	Round
⌘	41	10.00mil (0.254mm)	-	-	PTH	Round
⊞	233	10.00mil (0.254mm)	-	-	PTH	Round
	376 Total					

Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
Hole Length = Routed Path Length + Tool Size + Slot length as defined in the PCB layout



Drill Guide For (Bottom Layer,Top Layer)

A

MAT'L					
INTERPRET DRAWING PER ANSI Y32		DRAWN BY/DATE Liam Gallagher 13/2021	WWW.SWARM.SPACE SpaceX		
UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS IN INCHES.		CHECKED BY/DATE	DRAWING TYPE: Fabrication Drawing.		
TOLERANCE	.XX ± .01 .XXX ± .005	MFG ENG/DATE	Title: mPCle_Breakout01		
ANGLE	± 1°	FNL APPROVAL/DATE			
FRACTION	±1/64				
CONFIDENTIAL THIS DOCUMENT IS THE CONFIDENTIAL PROPERTY OF Swarm Technologies AND IS NOT TO BE REPRODUCED, USED OR DISCLOSED BY ANYONE WITHOUT THE EXPRESS WRITTEN PERMISSION OF THE Swarm Technologies COMPANY.		FNL APPROVAL/DATE	SIZE DWG No. mPCle_Breakout01	REV A1	
		PROD CTRL/FNL REL	B	SCALE: 1:1	SHEET: 1 OF 1