



Symbol	Hit Count	Finished Hole Size	Plated	Drill Layer Pair	Via/Pad	Hole Type
J	54	0.300mm <11.81mil	PTH	Top Layer - Bottom Layer	Via	Round
A	2	0.350mm <13.78mil	PTH	Top Layer - Bottom Layer	Via	Round
B	2	0.600mm <23.62mil	PTH	Top Layer - Bottom Layer	Pad	Round
C	30	0.800mm <31.50mil	PTH	Top Layer - Bottom Layer	Pad	Round
D	9	0.900mm <35.43mil	PTH	Top Layer - Bottom Layer	Pad	Round
E	20	1.000mm <39.37mil	PTH	Top Layer - Bottom Layer	Pad	Round
F	2	1.100mm <43.31mil	PTH	Top Layer - Bottom Layer	Pad	Round
G	4	1.600mm <62.99mil	PTH	Top Layer - Bottom Layer	Pad	Round
H	4	3.000mm <118.11mil	PTH	Top Layer - Bottom Layer	Pad	Round
I	4	3.200mm <125.98mil	NPTH	Top Layer - Bottom Layer	Pad	Round
	131 Total					

ASSEMBLY NOTES:

1. ALL COMPONENTS SHALL BE RoHS COMPLIANT.
2. ALL UNUSED THROUGH HOLE COMPONENT LOCATIONS SHALL BE FREE OF SOLDER.
3. ALL COMPONENTS SHALL BE MOUNTED FLUSH TO THE BOARD, EXCEPT AS NOTED.
4. FINISHED BOARD SHALL BE FREE OF ALL RESIDUES.
5. ALL LEADS SHALL BE TRIMMED TO A MAXIMUM HEIGHT OF 2mm

REV	ECO	Comments	Date

THIS PCB TO BE MANUFACTURED TO MEET ALL ACCEPTANCE LEVELS
OF A CLASS 2 PCB PER ANSI/IPC-A-600G.

MATERIAL:	FR-4 or Equivalent			
	<input checked="" type="checkbox"/> MULTILAYER	<input type="checkbox"/> 4 LAYERS	<input type="checkbox"/> CONTROLLED IMPEDANCE	
	Cu WEIGHT EXTERNAL LAYERS	<input type="checkbox"/> 70um	<input type="checkbox"/> FINISHED	
	Cu WEIGHT INTERNAL LAYERS	<input type="checkbox"/> 35um	<input type="checkbox"/> FINISHED	
	FINISHED OVERALL THICKNESS	<input type="checkbox"/> 1.6 mm	<input type="checkbox"/> ± <input type="checkbox"/> 10 %	
	COPPER THIEVING ALLOWED	<input type="checkbox"/> YES	<input checked="" type="checkbox"/> NO	

FINISH:

- ☐ LEAD—FREE HOT AIR LEVELING
- ☒ IMMERSION GOLD
- ☐ IMMERSION TIN
- ☐ SMOBC WITH SELECTIVE GOLD PLATING ON LANDS INDICATED. 1um GOLD OVER 5–10 um NICKEL

SOLDERMASK DYNACHEM EPIC 200 LPI OR EQUIVALENT

SOLDERMASK COLOR	GREEN HIGH GLOSS
------------------	------------------

SILKSCREEN COLOR WHITE

DRILLING: ☐ Blind / Buried Vias

Via Layer Pairs Top–Bottom

ALL HOLES TO BE LOCATED BY THE COORDINATES FROM THE NC DRILL DATA PROVIDED.

USE ARTWORK SET NO. 05-AS-17920

PCB REV 00

ALL UL LOGO, MANUFACTURER'S ID, AND DATE CODES SHALL BE PLACED ON THE BOTTOM SIDE UNLESS OTHERWISE INDICATED.

ANY ALTERNATIVES TO THE ABOVE SPECIFICATIONS MUST FIRST BE APPROVED.

Layer	Name	Material	Thickness	Constant	Board Layer Stack
	Top Overlay				
	Top Solder	Solder Resist	0.010mm	3.5	
1	Top Layer	Copper	0.070mm		
	Dielectric 1	PP-006	0.230mm	4.3	
2	signal	Copper	0.035mm		
	core	FR-4	0.930mm	4.3	
3	voltage	Copper	0.035mm		
	Dielectric 2	PP-006	0.230mm	4.3	
4	Bottom Layer	Copper	0.070mm		
	Bottom Solder	Solder Resist	0.010mm	3.5	
	Bottom Overlay				



TITLE:		PART NUMBER: SP6CA3	
SP6 1700V SiC Module Adapter Board			
PCB DESIGNER: Vipin G		GERBER FILE: 'Fab Drawing (Bottom,Top Layer)'	
ENGINEER: Nitesh S / Vipin G		BOARD NUMBER: 04-AS-17920	DOCUMENT NUMBER: 04-AS-17920-D DATE: 25-Aug-20
PCB FILE NAME: 04-AS-17920-RO.PcbDoc		LAYER NAME: FAB (M4)	REV: 00 10mm 400mil