

NOTES : UNLESS OTHERWISE SPECIFIED
 1. SPEC: PC-6012 CLASS 3 AND IPC600 CLASS 3.
 IN LOCATIONS WHERE DUE TO THE HIGH CIRCUIT DENSITY, WE HAVE REDUCED THE DESIGNED ANNULAR RING TO 5 MILS (0.125 MM),
 WE ONLY ALLOW TANGENCY OF THE VIAS TO THE PADS. WE DON'T ALLOW BREAKOUT.
 2. NUMBER OF LAYERS : 18

- 3. BOARD THICKNESS INCLUDING COPPER COATING : 1.93mm +/- 10%
- 4. MATERIAL: FR4
- a) IECQ IT180A or ISOLA 370HR, Tg = >170 Deg C, PER IPC-4101 / 26
- b) UL-94V-0 UNDERWRITERS LABORATORIES APPROVAL
- 5. LAYER STACK-UP: ACCORDING TO PDF FILES (ATTACHED TO ORDER)
- 6. COPPER FOIL IN ACCORDANCE WITH IPC-MF-150.
- 7. TWIST AND BOW <0.75%.

- 8. SOLDERMASK BOTH SIDES : ACCORDANCE WITH IPC-SM-840 CLASS-3.
 RECOMMENDED TAYO PSR 4000 OR TAMURA DSR2500 OR EQUIV.

SOLDER MASK THICKNESS SHOULD BE 15-25 MICRON.

- 9. FINISH: ENG IMMERSION GOLD OVER ELECTROLESS NICKEL PER IPC 4552.

10. ALL HOLES DIMENSIONS ARE AFTER PLATING.

11. ALL VIA DRILL SHOULD PASS FILLING AND CAPPING PROCESS. (ACCORDING TO DRILL TABLE BELOW).

12. SILKSCREEN : WHITE

13. The following information shall appear in silk screen:

a) UL-94V-0.

b) Manufacturer name or trademark.

c) Date code ww/yy

14. The PCB will be completely RoHS compliant as defined by Directive 2011/65/EC of the European Parliament.

15. EACH PRODUCTION BATCH / LOT SHOULD BE DELIVERED WITH C.O.C. TEST REPORT AND TEST COUPON PER LOT.

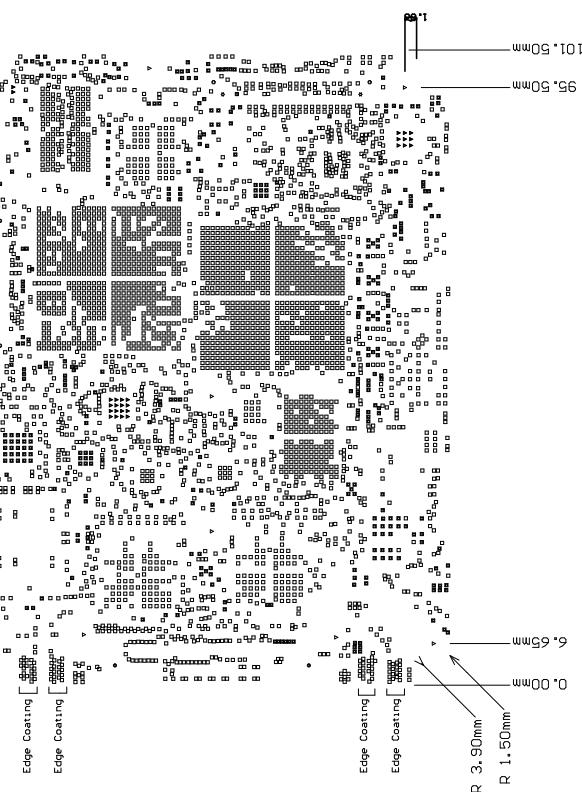
16. Electrical test: continuity and insulation at 250V 100% on all board shall be performed according to the Netlist.

17. The Gerber files shall be compared to the Netlist.

18. PACKAGING SHOULD BE ACCORDING TO 1601 A.

19. ADD TWO MARGINS 5mm WIDTH TO PCB BOARD.

DRILL



Symbol	Hole Size	Count	Plated	U1a/Pad	Hole Tolerance
□	0.20mm	3815	PTH	U1a	+0.05mm/-0.10mm
■	0.25mm	372	PTH	U1a	+0.05mm/-0.10mm
▼	0.30mm	27	PTH	Pad	+0.05mm/-0.08mm
✚	1.00mm	4	PTH	Pad	+/-0.08mm
◎	1.80mm	2	PTH	Pad	+/-0.08mm
▽	2.30mm	9	PTH	Pad	+/-0.08mm
✖	1.50mm	4	NPTH	Pad	+/-0.05mm
4233 Total					

COMMTRACT		
COMPANY:	DIGITAL PHOENIX	DATE:
PROJECT:	14-Feb-24	
LAYER :	PCB9550227-D	

NOTES : UNLESS OTHERWISE SPECIFIED

1. SPEC: IPC-6012 CLASS 3 AND IPC600 CLASS 3.
IN LOCATIONS WHERE DUE TO THE HIGH CIRCUIT DENSITY, WE HAVE REDUCED THE DESIGNED ANNULAR RING TO 5 MILS (0.125 MM),
WE ONLY ALLOW TANGENCY OF THE VIAS TO THE PADS, WE DON'T ALLOW BREAKOUT.

2. NUMBER OF LAYERS : 18

3. BOARD THICKNESS INCLUDING COPPER COATING : 1.95mm +/-10%

4. MATERIAL: FR4

a) ITO/N80A or ISOLA 370HR, Tg =>170 Deg C, PER IPC4101/126

b) UL-94V-0 UNDERWRITERS LABORATORIES APPROVAL

5. LAYER STACK-UP: ACCORDING TO PDF FILES (ATTACHED TO ORDER)

6. COPPER FOIL IN ACCORDANCE WITH IPC-MF-150.

7. TWIST AND BOW <0.75%

8. SOLDERMASK BOTH SIDES : ACCORDANCE WITH IPC-SM-840 CLASS-3.
RECOMMENDED TAYO PSR 4000 OR TAMURA DSR2500 OR EQUIV.

SOLDER MASK THICKNESS SHOULD BE 15-25 MICRON.

9. FINISH: ENIG IMMERSION GOLD OVER ELECTROLESS NICKEL PER IPC 4552.

10. ALL HOLES DIMENSIONS ARE AFTER PLATING.

11. ALL VIA DRILL SHOULD PASS FILLING AND CAPPING PROCESS. (ACCORDING TO DRILL TABLE BELOW).

12. SILKSCREEN : WHITE

13. The following information shall appear in silk screen:
a) UL-94V-0
b) Manufacturer name or trademark.
c) Date code ww/yy

14. The PCB will be completely RoHS compliant as defined by
Directive 2011/65/EC of the European Parliament.
15. EACH PRODUCTION BATCH / LOT SHOULD BE DELIVERED WITH C.O.C. TEST
REPORT AND TEST COUPON PER LOT.

16. Electrical test: continuity and insulation at 250V 100% on
all board shall be performed according to the Netlist.

17. The Gerber files shall be compared to the Netlist.

18. PACKAGING SHOULD BE ACCORDING TO 1601 A.

19. ADD TWO MARGINS 5mm WIDTH TO PCB BOARD.

