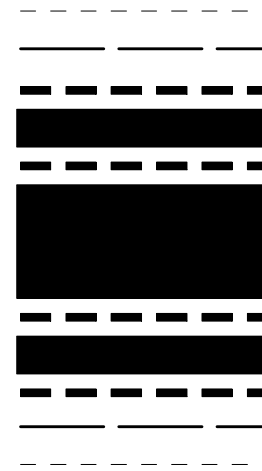


Drill Map:

- 0.20mm / 0.008" (479 holes)
- 0.30mm / 0.012" (27 holes)
- 0.33mm / 0.013" (2 holes)
- 0.60mm / 0.024" (0 holes + 4 slots)
- 0.65mm / 0.026" (0 holes + 2 slots)
- 0.70mm / 0.028" (0 holes + 2 slots)
- * 0.80mm / 0.031" (0 holes + 3 slots)
- 1.00mm / 0.039" (42 holes)
- ⊗ 4.20mm / 0.165" (4 holes)
- 0.50mm / 0.020" (42 holes) (not plated)
- 0.60mm / 0.024" (2 holes) (not plated)
- 0.70mm / 0.028" (1 hole) (not plated)
- 0.90mm / 0.035" (1 hole) (not plated)
- 0.99mm / 0.039" (3 holes) (not plated)
- ⊗ 2.75mm / 0.108" (4 holes) (not plated)

NOTES: UNLESS OTHERWISE SPECIFIED.

1. STANDARDS:
 - A. FABRICATE PCB IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-6012, CLASS 2.
 - B. INTERPRET DIMENSIONS AND TOLERANCES IN ACCORDANCE WITH THE CURRENT REVISION OF ASME Y14.5M.
 - C. DO NOT SCALE DRAWING.
2. MATERIAL:
 - A. CORE MATERIAL: FR4 To 180 C OR EQUIVALENT, PREPREG MATERIAL 2-2313 OR EQUIVALENT.
 - B. EQUIVALENT MATERIAL SHALL BE ROHS COMPLIANT, HALOGEN FREE AND APPROVED BY YourCo.
 - C. THICKNESS OF INDIVIDUAL COPPER CLAD SHEETS SHALL BE IN AS DEFINED IN STACK-UP.
3. FLATNESS:
 - A. BOW AND TWIST OF ASSEMBLY SUB-PANEL OR SINGULATED PWB SHALL NOT EXCEED .025 MM PER MM.
 - B. TEST IN ACCORDANCE WITH THE CURRENT REVISION OF IPC-TM-650 2.4.22
4. ETCH GEOMETRY:
 - A. MEASURE WIDTH FROM THE BASE OF THE METALLIZATION.
 - B. MINIMUM LINE WIDTH: 0.09 MM OUTER, 0.09 MM INNER LAYERS.
 - C. FINISHED LINE WIDTH AND TERMINAL AREA SHALL NOT DEVIATE FROM THE 1-TO-1 MASTER PATTERN IMAGE BY MORE THAN +/- 0.025 MM OR 20%, WHICHEVER IS LESS.
5. SURFACE FINISH: (SELECT APPROPRIATE FINISH(ES))
 - A. ENIG PLATING PER CURRENT REVISION OF IPC-4552. EXPOSED METAL SHALL HAVE 118-236 MICRO INCHES ELECTROLESS NICKEL AND 2-5 MICRO INCHES GOLD.
6. DESTRUCTIVE TESTING:
 - A. MICRO SECTION SAMPLE AND REPORT SHALL BE PROVIDED TO ANALOG LIFE LLC.
 - B. X-OUT PANELS MAY BE USED FOR SOLDER SAMPLE.
7. HOLES:
 - A. PLATING IN HOLES SHALL BE CONTINUOUS ELECTROLYTIC COPPER WITH 0.025 MM MINIMUM BARREL THICKNESS.
 - B. MINIMUM FINISHED HOLE SIZE: 0.2 MM
 - C. HOLE SIZE MEASURED AFTER PLATING.
 - D. SEE DRILL CHART FOR FINISHED HOLE SIZE AND TOLERANCE.
 - E. ALL HOLES SHALL BE LOCATED WITHIN 0.05 MM OF TRUE POSITION AS SUPPLIED IN CAD DATA.
8. SOLDERMASK:
 - A. SOLDERMASK OVER BARE COPPER (SMOBC) ON PRIMARY AND SECONDARY SIDES USING SUPPLIED ARTWORK IN ACCORDANCE WITH CURRENT REVISION OF IPC-SM-840 TYPE B.
 - B. COLOR: GREEN
 - C. LIQUID PHOTO-IMAGEABLE (LPI) 0.001 MM TO 0.002 MM THICKNESS, HALOGEN FREE
 - D. NO BLEED-OUT ALLOWED OVER EXPOSED SMD PADS.
 - E. NO EXPOSED TRACES.
9. SILKSCREEN:
 - A. SILKSCREEN PRIMARY AND SECONDARY SIDE WITH WHITE EPOXY, NON-CONDUCTIVE, NON-NUTRIENT INK.
 - B. ANY UNSPECIFIED STROKE WIDTH SHALL BE 0.13 MM
 - C. CLIP SILKSCREEN AWAY FROM ANY EXPOSED METAL.
 - D. VENDOR DATE CODE, LOGO, UL AND ANY ADDITIONAL MARKING TO BE LOCATED ON THE SECONDARY SIDE.
 - E. BAG AND TAG ACCEPTABLE FOR PWBs THAT ARE TOO SMALL FOR MARKING.
10. REMOVE ALL BURRS AND BREAK SHARP EDGES R0.003 MIN.
11. NON-DESTRUCTIVE EVALUATION:
 - A. ALL PWBs SHALL PASS 100% ELECTRICAL TEST USING SUPPLIED IPC-356 NETLIST IN ACCORDANCE WITH CURRENT REVISION OF IPC-9252, CLASS 2.
 - B. CERTIFICATE OF CONFORMANCE SHALL BE SUPPLIED WITH EACH SHIPMENT.
12. X-OUTS:
 - A. X-OUT BOARDS THAT DO NOT MEET ALL SPECIFICATIONS USING PERMANENT MARKING ON BOTH SIDES OF THE AFFECTED PCB.
 - B. PANELS THAT DO NOT HAVE ANY X-OUTS SHALL BE PACKAGED TOGETHER.
 - C. PANELS THAT HAVE 2 OR FEWER X-OUTS SHALL BE PACKAGED SEPARATE FROM NON-X-OUT PANELS.
 - D. PANELS WITH MORE THAN 2 X-OUTS SHALL BE REJECTED.
13. PACKAGING REQUIREMENTS:
 - A. PWBs SHALL BE PACKAGED IN VACUUM SEALED INNER CONTAINERS.
 - B. OUTER CONTAINERS SHALL BE SUFFICIENT TO PREVENT DAMAGE DURING SHIPPING AND HANDLING.
14. IMPEDANCE (ALL TOLERANCES +/- 10%)
 - A. ALL 0.2 MM WIDE TRACES ON OUTER LAYERS SHALL BE 50 OHMS.
 - B. ALL 0.1473 MM WIDE/0.1016 MM SPACE PAIRS ON OUTER LAYERS SHALL BE 90 OHMS.
 - D. VENDOR MAY ADJUST DESIGN GEOMETRIES UP TO +/-20% TO ACHIEVE TARGET IMPEDANCE. ADJUSTMENTS BEYOND 20% OF LINE WIDTH, SPACING OR DIELECTRIC THICKNESS SHALL REQUIRE APPROVAL FROM ANALOG LIFE LLC.



Layer (Material) Type	Layer (File) Name	Thickness (mm)	Dielectric Constant	Notes
Silkscreen	F.Silk	0.00762		
Soldermask	F.Mask	0.0127	3.8	
Copper	F.Cu	0.035		
2-2313 (Prepreg)	-----	0.1	4.05	
Copper	PWR	0.0175		
FR4 (Core)	-----	1.265		
Copper	GND	0.0175		GND on layer 3 because it is referenced to bottom components for RF
2-2313 (Prepreg)	-----	0.1	4.05	
Copper	B.Cu	0.035		
Soldermask	B.Mask	0.0127	3.8	
Silkscreen	B.Silk	0.00762		



Drawn by Analog Life LLC
Contextual Electronics
Sheet:
File: Advanced-BLE-CELL.kicad_pcb

Title: ABC

Size: B Date: 2020-08-07
KiCad E.D.A. kicad (5.1.6)-1

Rev: A
Id: 1/1