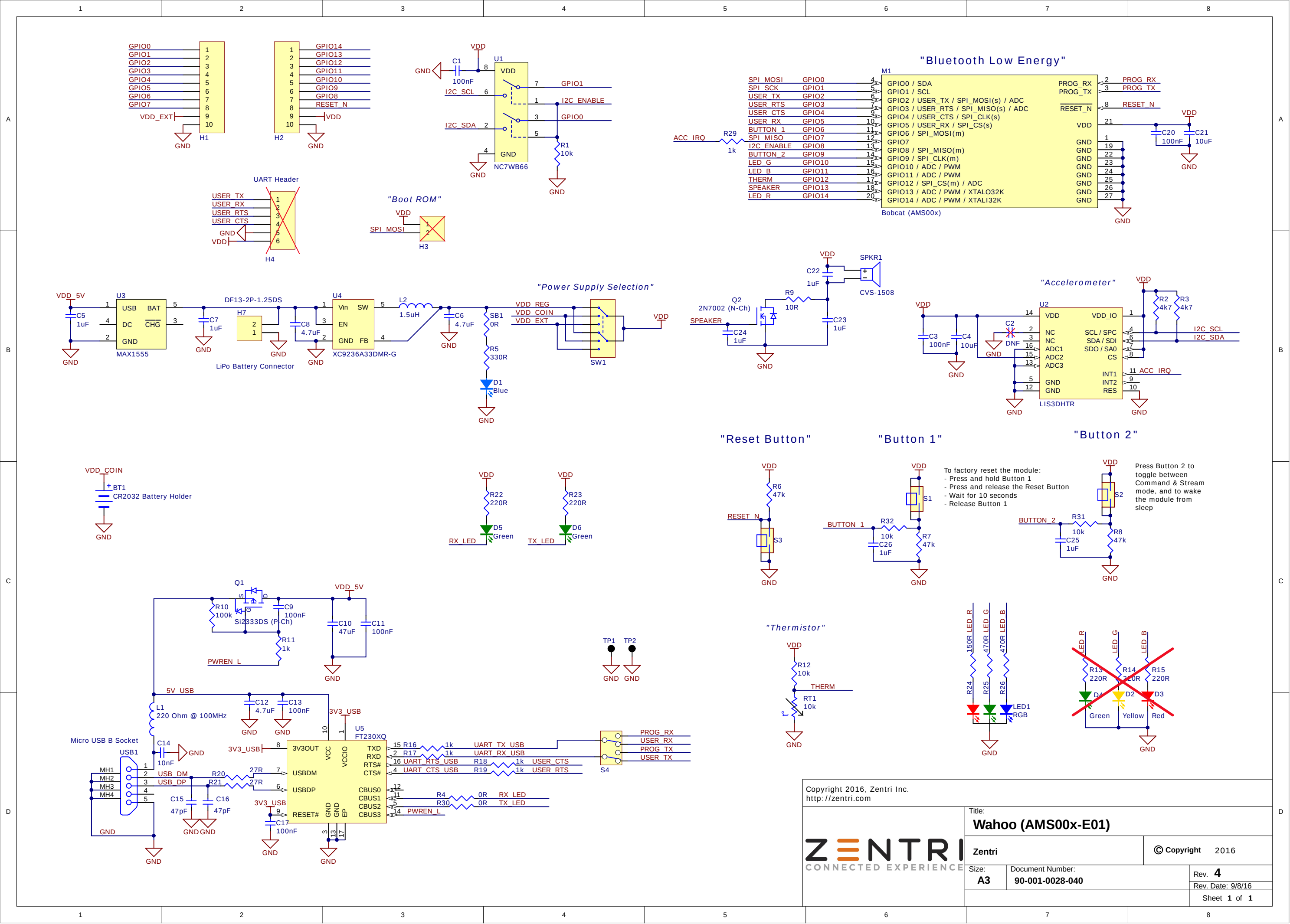
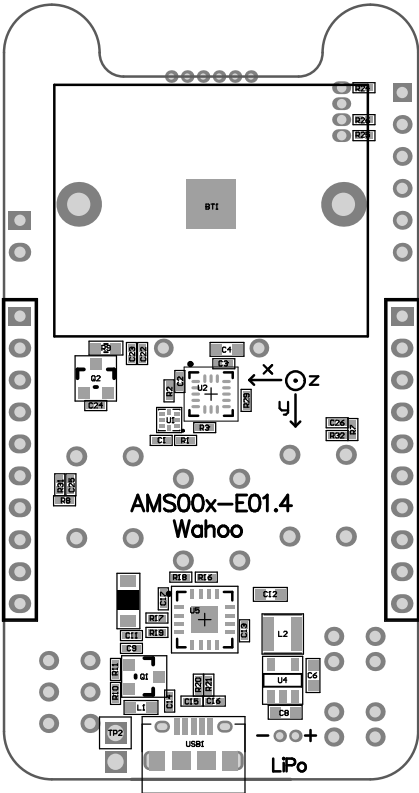
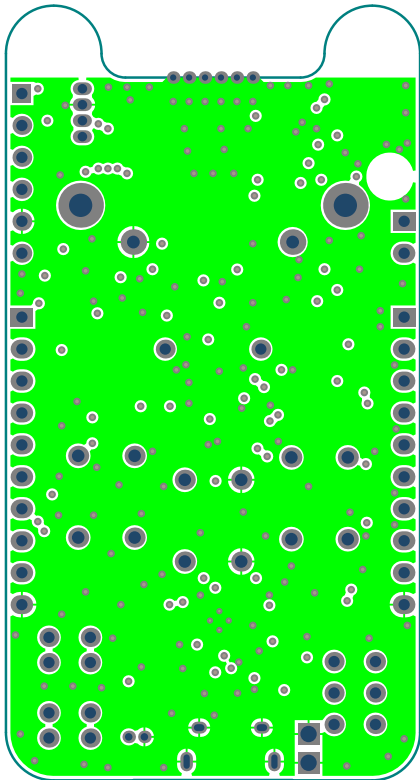


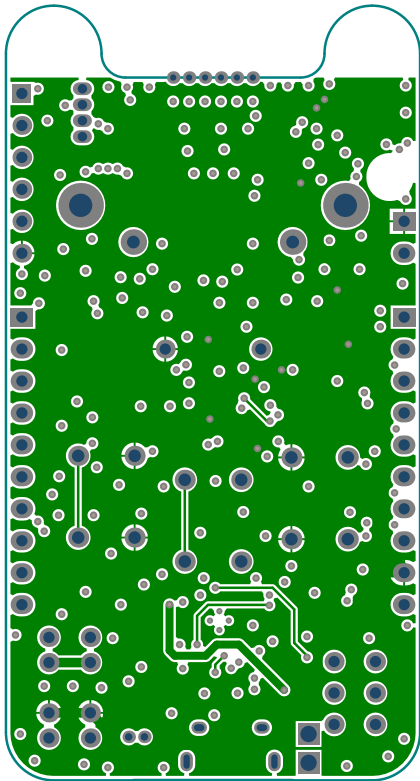




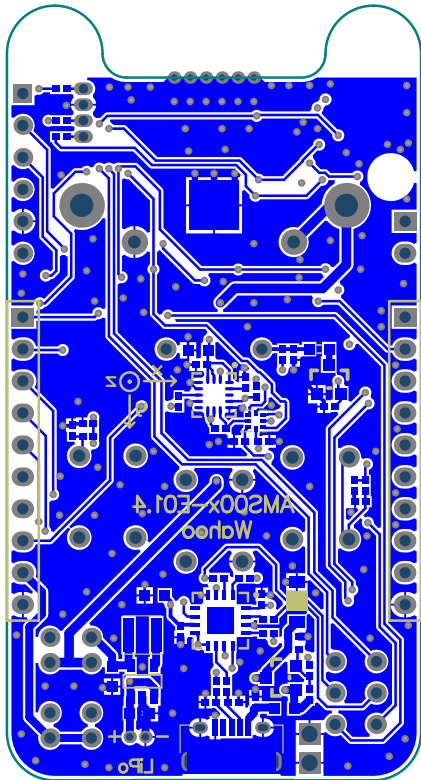




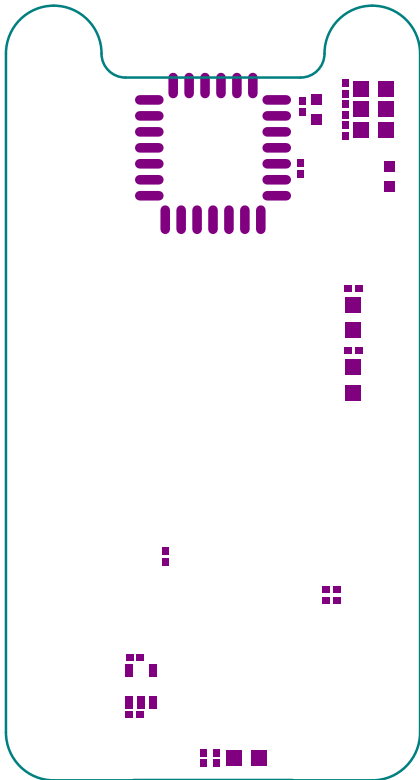
MidLayer1



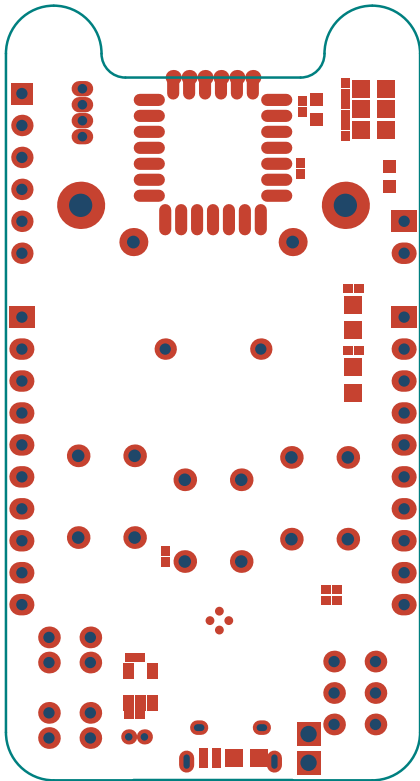
MidLayer2



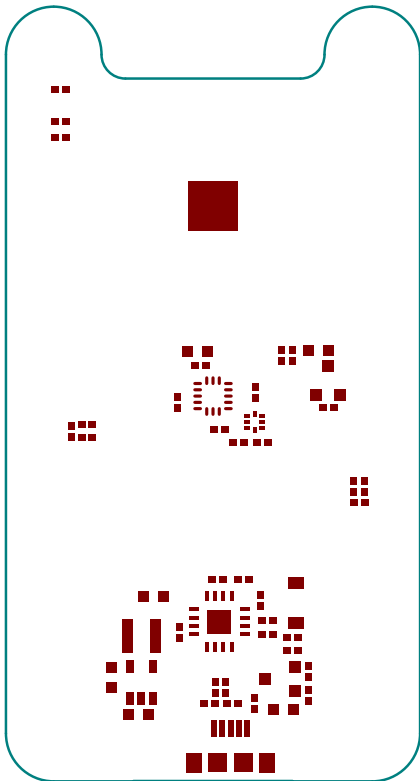
Bottom Layer



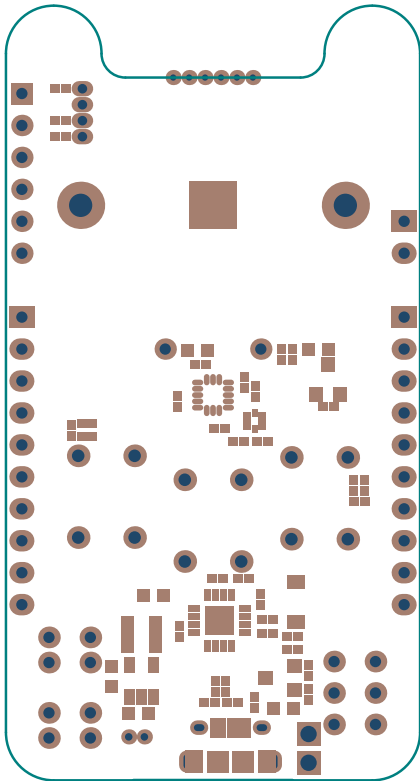
Top Paste



Top Solder



Bottom Paste



Bottom Solder

NOTES: UNLESS OTHERWISE SPECIFIED0

1. SPECIFICATIONS/TOLERANCES:
- A. FABRICATE PER IPC-6012, CLASS 2, USING PROVIDED DATA FILES XXXXXXXX.ZIP
 - B. ALL SPECIFICATIONS USED SHALL BE PER THEIR LATEST REVISIONS.
 - C. THE DIMENSIONS OF CIRCUIT FEATURES IN THE PROVIDED DATA MAY BE ADJUSTED ONLY TO COMPENSATE FOR PROCESS TOLERANCES; ADDING, REMOVING OR RELOCATING CIRCUIT FEATURES IS NOT ALLOWED
 - D. REMOVE ALL BURRS AND BREAK SHARP EDGES, .381 [.015] MAX RADIUS.
 - E. PARENTHETICAL INFORMATION IS FOR REFERENCE ONLY.
 - F. REPAIR OF PCB DEFECTS IS NOT PERMITTED.

2. DIELECTRIC MATERIAL:
- A. DIELECTRIC MATERIAL SHALL BE PER IPC-4101/99, /124, /126 OR /129 (RoHS COMPLIANT EPOXY-GLASS)
 - B. MINIMUM DIELECTRIC THICKNESS SHALL BE .051 [.002] FOR REFERENCED STACK-UP DIMENSIONS OF.076 [.003] OR GREATER; IPC-6012 REQUIREMENTS SHALL OTHERWISE APPLY. SINGLE-PLY CONSTRUCTION IS ALLOWED.
 - C. SEE LAYER STACK-UP FOR REQUIRED COPPER WEIGHTS AND THE FINISHED PCB THICKNESS. IF SPECIFIED, 1/3 OZ. STARTING FOIL MAY BE ACHIEVED BY 1/2 OZ. FOIL REDUCTION.
 - D. FINISHED PCB THICKNESS SHALL BE MEASURED OVER LANDS AND/OR CONDUCTORS NOT COVERED BY SOLDER MASK. IF SPECIFIED, 1/3 OZ. STARTING FOIL MAY BE ACHIEVED BY 1/2 OZ. FOIL REDUCTION.

3. DRILLING:
- A. VIA DIAMETERS SHALL BE VERIFIED BEFORE PLATING; ALL OTHER HOLE DIAMETERS ARE FINISHED DIMENSIONS.
 - B. LAYER-TO-LAYER MISREGISTRATION SHALL BE .127 [.005] MAXIMUM.

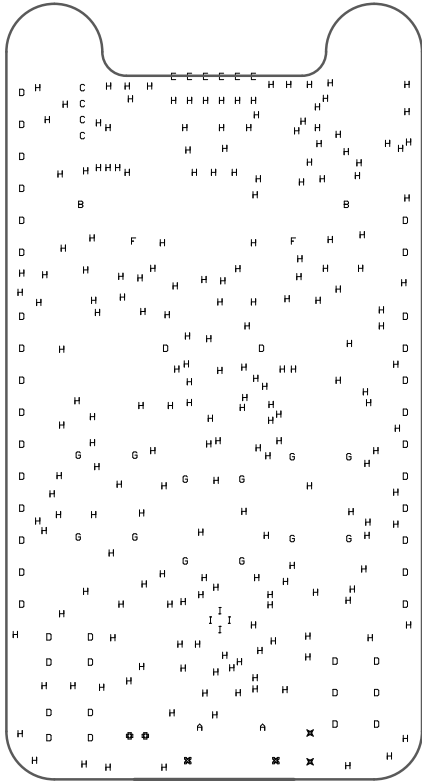
4. SOLDER MASK:
- A. APPLY LPI SOLDER MASK OVER BARE COPPER USING PROVIDED DATA.
 - B. SOLDER MASK SHALL BE PER IPC-SM-840, CLASS T, COLOR ORANGE.
 - C. THE DIMENSIONS OF SOLDER MASK-DEFINED PADS ON PLANES AND/OR WIDE CONDUCTORS SHALL NOT BE MODIFIED.

5. SILKSCREEN/MARKING:
- A. SILKSCREEN PCB PER PROVIDED DATA USING PERMANENT, NON-CONDUCTIVE INK, COLOR WHITE.
 - B. SUPPLIER ID AND TRACEABILITY INFORMATION SHALL NOT BE APPLIED ON ANY PART OF THE PCB.
 - C. INK SHALL NOT BE APPLIED TO ANY SOLDERABLE SURFACE. USING PROVIDED DATA.

6. ELECTRICAL TEST:
- A. DESIGN VERIFICATION SHALL BE DONE PRIOR TO PCB FABRICATION USING GERBER DATA AND AN IPC-D-356 NETLIST.
 - B. ALL PCBs SHALL BE 100% ELECTRICALLY TESTED FOR OPENS AND SHORTS
 - C. APPLY TEST STAMP IN NON-LEGEND AREA ON SECONDARY SIDE OF PCB.

7. FINAL FINISH:
- A. FINAL FINISH SHALL BE ELECTROLESS NICKEL/IMMERSION GOLD (ENIG) PER IPC-4552.

9. IF PANELIZATION SPECIFICATIONS ARE PROVIDED, THE PCBs SHALL BE DELIVERED IN PANEL FORM. HOWEVER, THESE SPECIFICATIONS MAY BE REPLACED BY SUPPLIER PREFERRED (SPECIFICATIONS AS APPROVED).



Drill Drawing

	Project: Wahoo	
	Company: Zentri	
	Drawn By: Chad O'Neill	
	PCB Num: 90-002-0028-040	
	Date: 9/8/16	Rev: 4

Symbol	Hit Count	Finished Hole Size	Plated	Hole Type
⌘	2	0.500mm (19.69mil)	PTH	Slot
A	2	0.550mm (21.65mil)	PTH	Slot
⊕	2	0.600mm (23.62mil)	PTH	Round
F	2	1.016mm (40.00mil)	PTH	Round
✕	2	1.300mm (51.18mil)	PTH	Round
B	2	1.854mm (73.00mil)	PTH	Round
I	4	0.200mm (7.87mil)	PTH	Round
C	4	0.762mm (30.00mil)	PTH	Round
E	6	0.500mm (19.69mil)	PTH	Round
G	12	1.000mm (39.37mil)	PTH	Round
D	44	0.900mm (35.43mil)	PTH	Round
H	206	0.300mm (11.81mil)	PTH	Round
	288 Total			

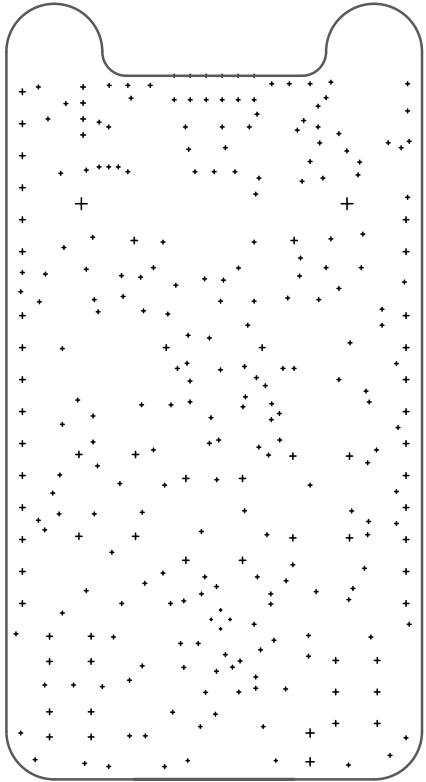
Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
Hole Length = Routed Path Length + Tool Size = Slot length as defined in the PCB layout

BOARD STACKUP		LAYER DESCRIPTION	START COPPER WT
62 MIL +/- 10%	X MIL	TOP LAYER	0.5 OZ
	X MIL	MID LAYER 1	1.0 OZ
	X MIL	MID LAYER 2	1.0 OZ
		BOTTOM LAYER	0.5 OZ

FILE EXTENSIONS:

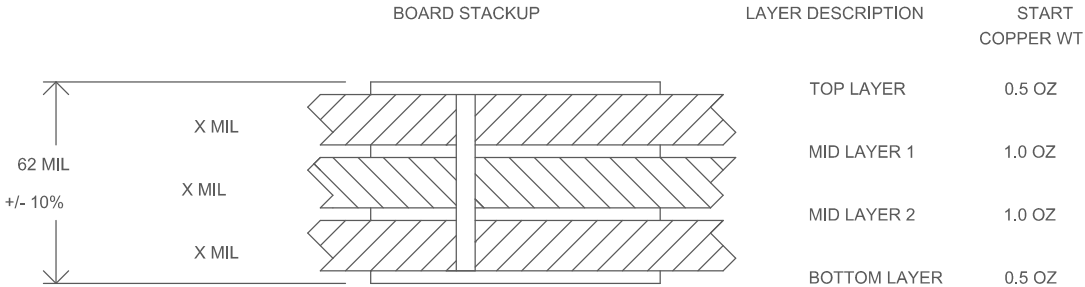
- *.GTO - Top Overlay (Silkscreen)
- *.GTS - Top Solder Mask
- *.GTP - Top Solder Paste
- *.GBS - Bottom Solder Mask
- *.GD1 - Drill Drawing
- *.GG1 - Drill Guide
- *.GKO - Keep Out
- *.TXT - NC Drill Top-Bottom

- NOTES: UNLESS OTHERWISE SPECIFIED0
1. SPECIFICATIONS/TOLERANCES:
- A. FABRICATE PER IPC-6012, CLASS 2, USING PROVIDED DATA FILES XXXXXXXX.ZIP
 - B. ALL SPECIFICATIONS USED SHALL BE PER THEIR LATEST REVISIONS.
 - C. THE DIMENSIONS OF CIRCUIT FEATURES IN THE PROVIDED DATA MAY BE ADJUSTED ONLY TO COMPENSATE FOR PROCESS TOLERANCES; ADDING, REMOVING OR RELOCATING CIRCUIT FEATURES IS NOT ALLOWED
 - D. REMOVE ALL BURRS AND BREAK SHARP EDGES, .381 [.015] MAX RADIUS.
 - E. PARENTHETICAL INFORMATION IS FOR REFERENCE ONLY.
 - F. REPAIR OF PCB DEFECTS IS NOT PERMITTED.
2. DIELECTRIC MATERIAL:
- A. DIELECTRIC MATERIAL SHALL BE PER IPC-4101/99, /124, /126 OR /129 (RoHS COMPLIANT EPOXY-GLASS)
 - B. MINIMUM DIELECTRIC THICKNESS SHALL BE .051 [.002] FOR REFERENCED STACK-UP DIMENSIONS OF.076 [.003] OR GREATER; IPC-6012 REQUIREMENTS SHALL OTHERWISE APPLY. SINGLE-PLY CONSTRUCTION IS ALLOWED.
 - C. SEE LAYER STACK-UP FOR REQUIRED COPPER WEIGHTS AND THE FINISHED PCB THICKNESS. IF SPECIFIED, 1/3 OZ. STARTING FOIL MAY BE ACHIEVED BY 1/2 OZ. FOIL REDUCTION.
 - D. FINISHED PCB THICKNESS SHALL BE MEASURED OVER LANDS AND/OR CONDUCTORS NOT COVERED BY SOLDER MASK. IF SPECIFIED, 1/3 OZ. STARTING FOIL MAY BE ACHIEVED BY 1/2 OZ. FOIL REDUCTION.
3. DRILLING:
- A. VIA DIAMETERS SHALL BE VERIFIED BEFORE PLATING; ALL OTHER HOLE DIAMETERS ARE FINISHED DIMENSIONS.
 - B. LAYER-TO-LAYER MISREGISTRATION SHALL BE .127 [.005] MAXIMUM.
4. SOLDER MASK:
- A. APPLY LPI SOLDER MASK OVER BARE COPPER USING PROVIDED DATA.
 - B. SOLDER MASK SHALL BE PER IPC-SM-840, CLASS T, COLOR ORANGE.
 - C. THE DIMENSIONS OF SOLDER MASK-DEFINED PADS ON PLANES AND/OR WIDE CONDUCTORS SHALL NOT BE MODIFIED.
5. SILKSCREEN/MARKING:
- A. SILKSCREEN PCB PER PROVIDED DATA USING PERMANENT, NON-CONDUCTIVE INK, COLOR WHITE.
 - B. SUPPLIER ID AND TRACEABILITY INFORMATION SHALL NOT BE APPLIED ON ANY PART OF THE PCB.
 - C. INK SHALL NOT BE APPLIED TO ANY SOLDERABLE SURFACE. USING PROVIDED DATA.
6. ELECTRICAL TEST:
- A. DESIGN VERIFICATION SHALL BE DONE PRIOR TO PCB FABRICATION USING GERBER DATA AND AN IPC-D-356 NETLIST.
 - B. ALL PCBs SHALL BE 100% ELECTRICALLY TESTED FOR OPENS AND SHORTS
 - C. APPLY TEST STAMP IN NON-LEGEND AREA ON SECONDARY SIDE OF PCB.
7. FINAL FINISH:
- A. FINAL FINISH SHALL BE ELECTROLESS NICKEL/IMMERSION GOLD (ENIG) PER IPC-4552.
9. IF PANELIZATION SPECIFICATIONS ARE PROVIDED, THE PCBs SHALL BE DELIVERED IN PANEL FORM. HOWEVER, THESE SPECIFICATIONS MAY BE REPLACED BY SUPPLIER PREFERRED (SPECIFICATIONS AS APPROVED).



Drill Guide

	Project: Wahoo	
	Company: Zentri	
	Drawn By: Chad O'Neill	
	PCB Num: 90-002-0028-040	
	Date: 9/8/16	Rev: 4



FILE EXTENSIONS:

- *.GTO - Top Overlay (Silkscreen)
- *.GTS - Top Solder Mask
- *.GTP - Top Solder Paste
- *.GBS - Bottom Solder Mask
- *.GD1 - Drill Drawing
- *.GG1 - Drill Guide
- *.GKO - Keep Out
- *.TXT - NC Drill Top-Bottom

