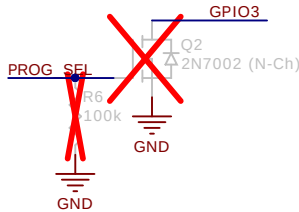
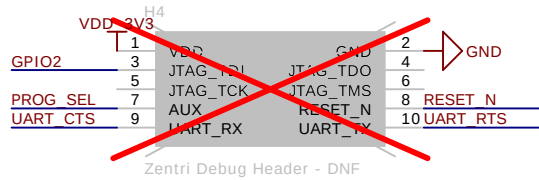
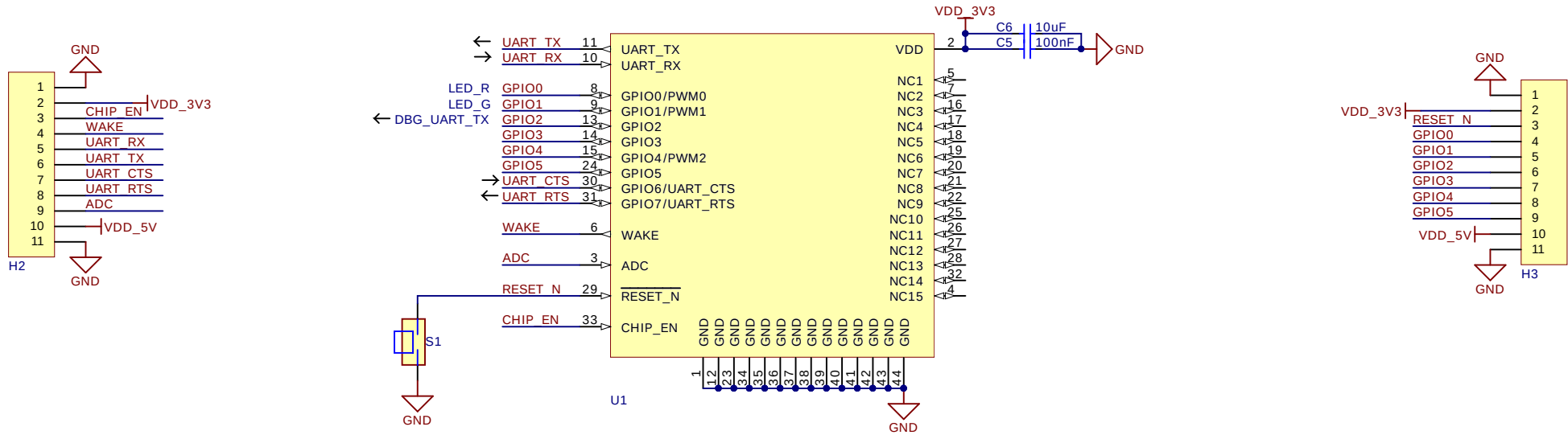
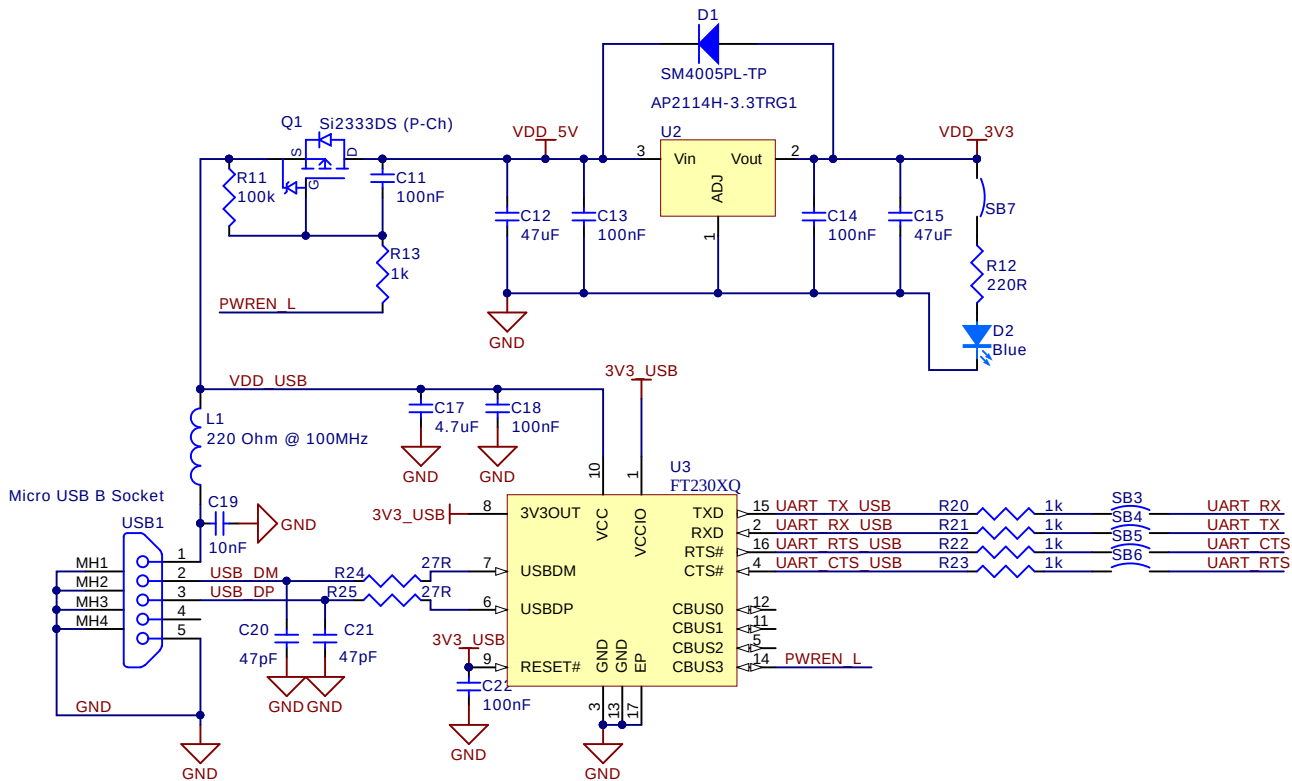
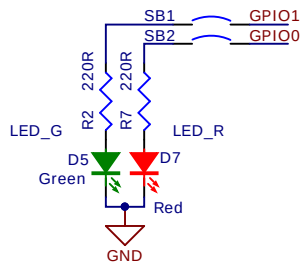


Blowfish Eval Board (AMW037-E01)



"User LED's"



Copyright 2016, Zentri Inc.
<http://zentri.com>

ZENTRI
CONNECTED EXPERIENCE

Title:
Blowfish Eval Board (AMW037-E01)

Zentri

© Copyright 2016

Size:
A3

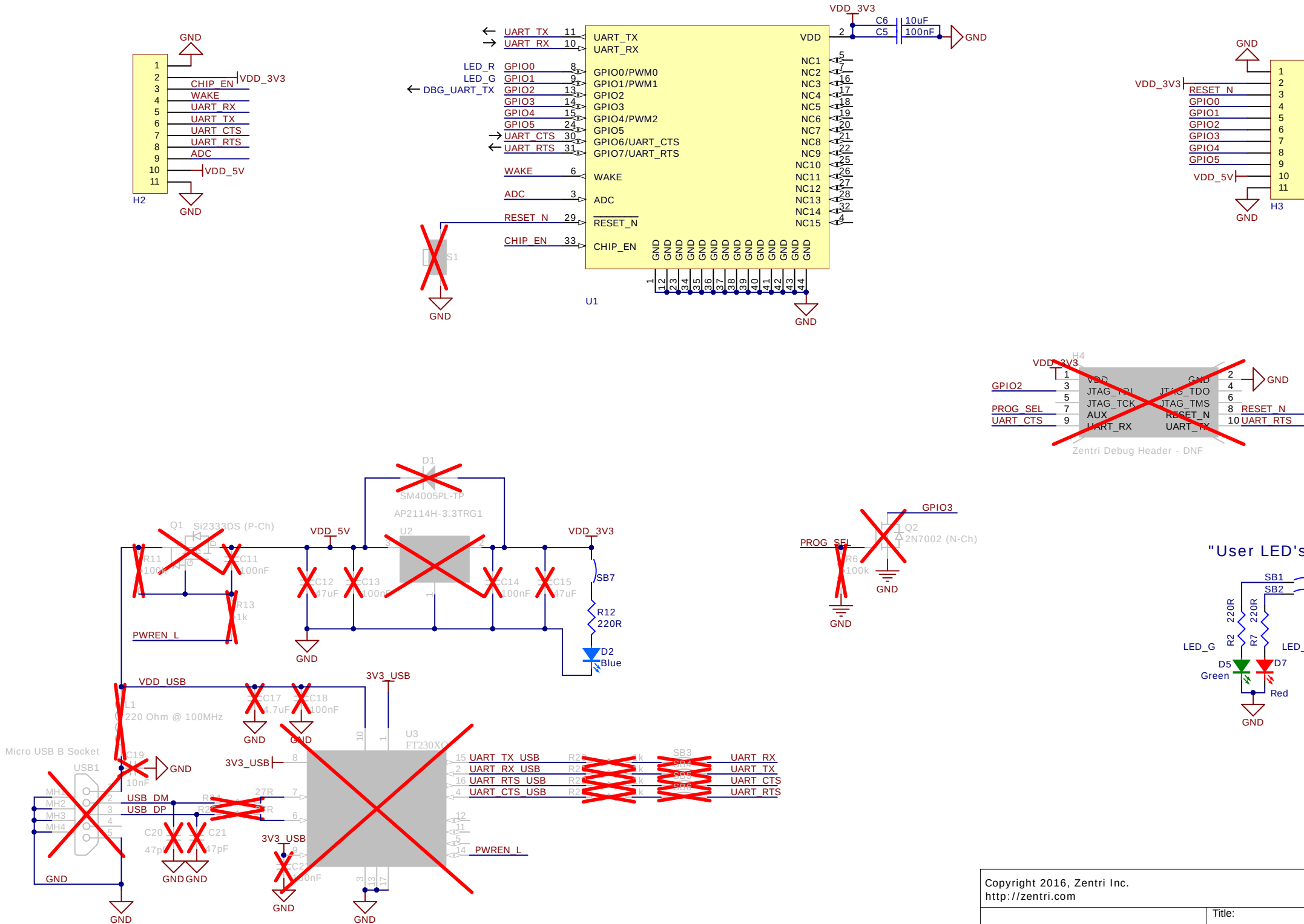
Document Number:

Rev. **3**

Rev. Date: 31/03/2017

Sheet **1** of **1**

Blowfish Adapter (AMW037-A1P)



Copyright 2016, Zentri Inc.
http://zentri.com

Title: **Blowfish Adapter (AMW037-A1P)**



Zentri

© Copyright 2016

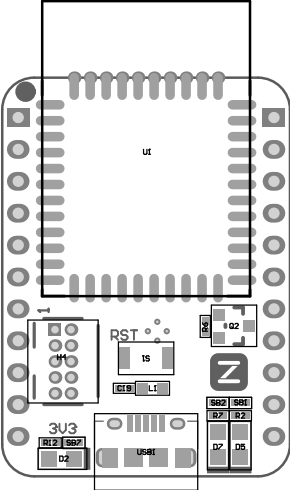
Size:
A3

Document Number:

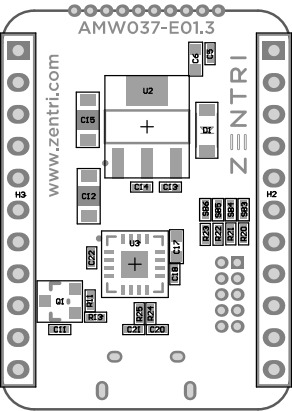
Rev. **3**

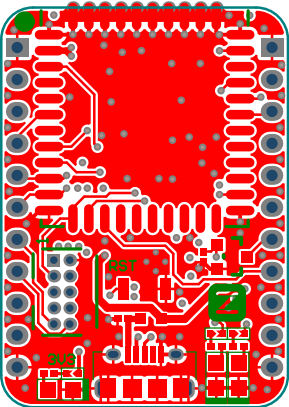
Rev. Date: 31/03/2017

Sheet 1 of 1

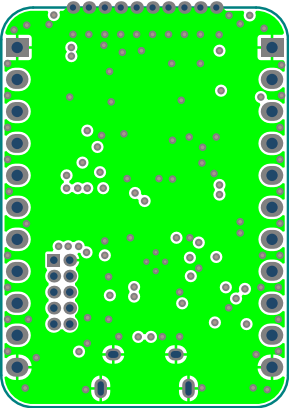


Top Overlay

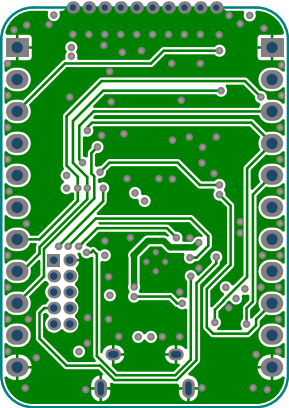




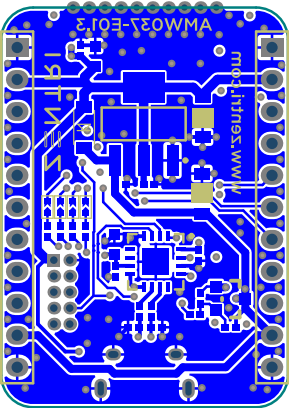
Top Overlay



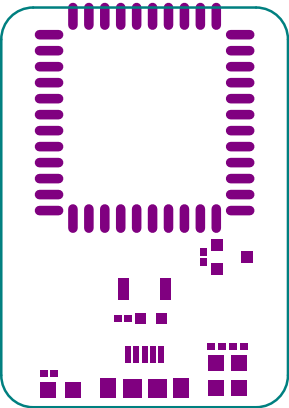
Ground Layer



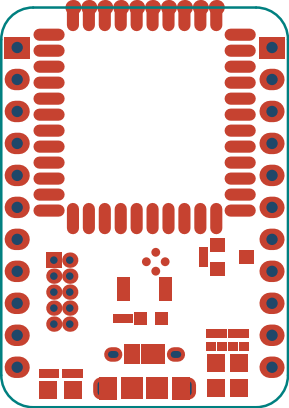
MidLayer1



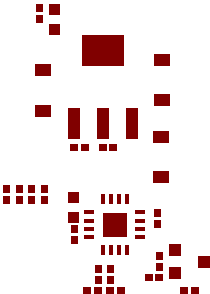
Bottom Layer



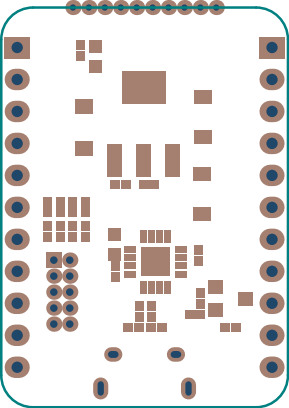
Top Paste



Top Solder



Bottom Paste



Bottom Solder

UNLESS OTHERWISE SPECIFIED:

1. SPECIFICATIONS/TOLERANCES:

- FABRICATE PER IPC-6012, CLASS 2, USING PROVIDED DATA FILES XXXXXXXX.ZIP
- ALL SPECIFICATIONS USED SHALL BE PER THEIR LATEST REVISIONS.
- 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
- REMOVE ALL BURRS AND BREAK SHARP EDGES, .381 [.015] MAX RADIUS.
- PARENTHETICAL INFORMATION IS FOR REFERENCE ONLY.
- REPAIR OF PCB DEFECTS IS NOT PERMITTED.

2. DIELECTRIC MATERIAL:

- DIELECTRIC MATERIAL SHALL BE PER IPC-4101/99, /124, /126 OR /129 (RoHS COMPLIANT EPOXY-GLASS)
- 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.
- 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.
- 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:

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

4. SOLDER MASK:

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

5. SILKSCREEN/MARKING:

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

6. ELECTRICAL TEST:

- DESIGN VERIFICATION SHALL BE DONE PRIOR TO PCB FABRICATION USING GERBER DATA AND AN IPC-D-356 NETLIST IF PROVIDED.
- ALL PCBs SHALL BE 100% ELECTRICALLY TESTED FOR OPENS AND SHORTS

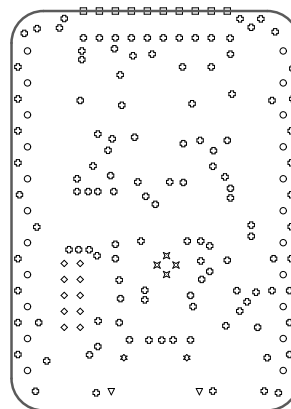
7. FINAL FINISH:

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

8. IMPEDANCE:

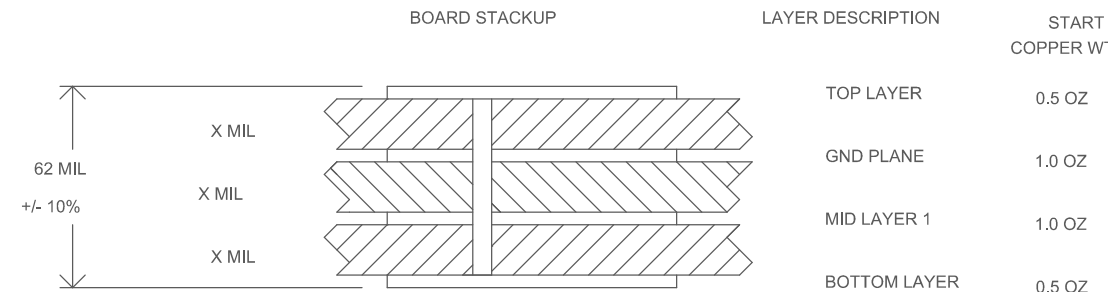
- IMPEDANCE TOLERANCE SHALL BE +/- 10%.
- SEE LAYER STACK-UP FOR IMPEDANCE REQUIREMENTS.

- 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

ZENTRI	Project: AMW037-E01	
	Company: Zentri	
	Drawn By: Chad O'Neill	
	Date: 31/03/2017	Rev: 3



FILE EXTENSIONS:

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

UNLESS OTHERWISE SPECIFIED:

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 MATT BLACK.
- 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 IF PROVIDED.
- B. ALL PCBs SHALL BE 100% ELECTRICALLY TESTED FOR OPENS AND SHORTS

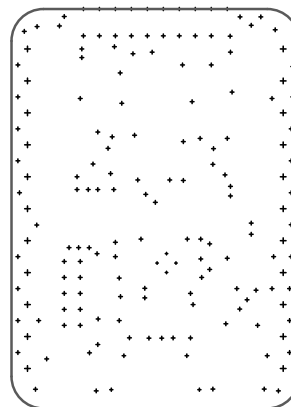
7. FINAL FINISH:

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

8. IMPEDANCE:

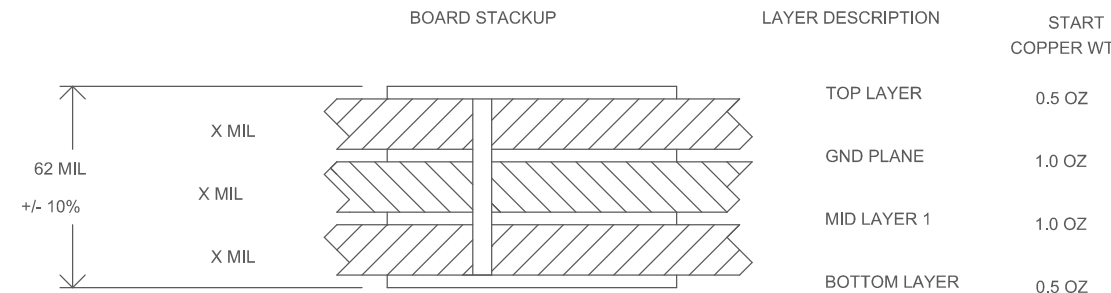
- A. IMPEDANCE TOLERANCE SHALL BE +/- 10%.
- B. SEE LAYER STACK-UP FOR IMPEDANCE REQUIREMENTS.

- 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).



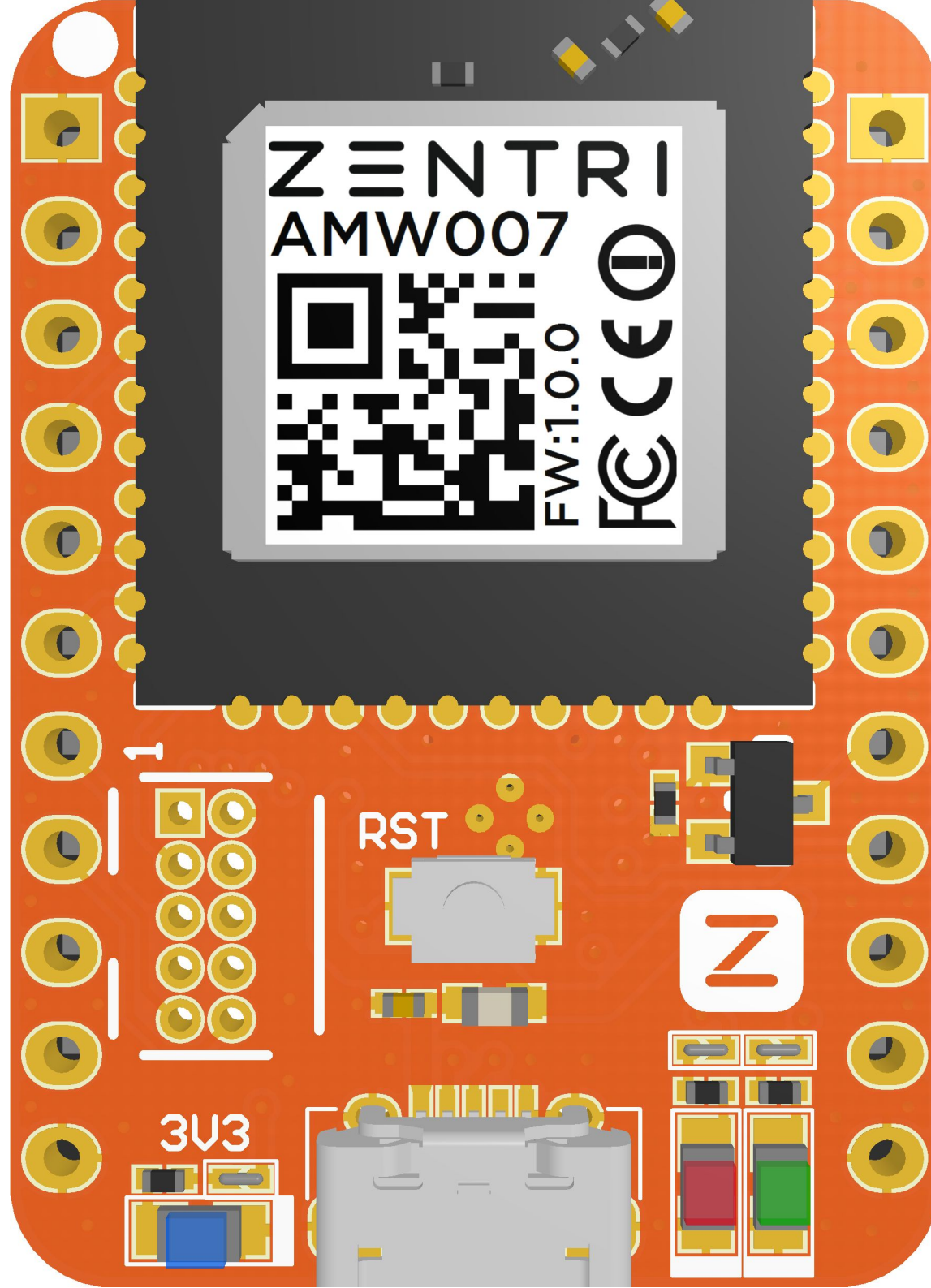
Drill Guide

ZENTRI	Project: AMW037-E01	
	Company: Zentri	
	Drawn By: Chad O'Neill	
	Date: 31/03/2017	Rev: 3



FILE EXTENSIONS:

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



ZENTRI

AMW007



FW:1.0.0



RST

3V3



AMW037-E01.3

www.zentri.com

ZENTRI

