PCB FABRICATION NOTES

BOARD NAME: BB-SPEMEVAL-A-1-NDAA

1) PCB TO BE MANUFACTURED IN ACCORDANCE WITH IPC6012, CURRENT REVISION, PERFORMANCE CLASS 2.

2) 4-LAYER RIGID PCB. MATERIAL FR4 0.062 THICK.

3) PROCESS: APPLY **RED** SOLDERMASK OVER BARE COPPER ON L1 (TOP) AND L4 (BOTTOM).

4) APPLY ENIG TO EXPOSED PADS ON, L1 AND L4, AFTER SOLDERMASK. MINIMUM THICKNESS PER IPC6012 CLASS 2.

5) SILKSCREEN COMPONENT LOCATION MARKINGS, WHITE PERMANENT, NON-CONDUCTIVE INK, L1, PER SUPPLIED GERBER FILE. NO SILKSCREEN ON L4.

6) MINIMUM COPPER IN PLATED HOLES PER IPC6012, CURRENT REVISION, CLASS 2.

7) ALL TOLERANCES PER IPC6012 CLASS 2.

8) FABRICATE AND INSPECT TO IPC CLASS 2 STANDARDS. PCB IS FOR INDUSTRIAL USE. IN CASE OF CONFLICT, SUPERCEDE IPC SPECIFICATION.

9) LAYER 1 IS TOP COPPER. LAYER SEQUENCE IS TOP / IN1 / IN2 / BOT (L1 / L2 / L3 / L4).

10) 100% ELECTRICAL TEST REQUIRED FOR SHORTS AND OPENS.

11) SUPPLY AS INDIVIDUAL PCBS.

12) IF PAD IS LARGER THAN NON-PLATED HOLE, DO NOT REMOVE PAD, AND DO NOT REMOVE ANY COPPER FROM PAD.

13) PCB USES VIA IN PAD. PLUG ALL VIAS IN PADS AND THEN PLATE WITH ENIG.

14) USE 1-OZ ON EXTERNAL LAYERS.

15) USE 1-OZ ON INTERNAL LAYERS.

16) IMPEDANCE REQUIREMENT: 100 OHM DIFFERENTIAL PAIR AND 50 OHM SINGLE ENDED AS IN ATTACHED FILES.