**002-0081 Rev – Manufacturing Notes:**

1. PCB
2. 0.062” thick
3. Copper on 4 layers.
4. Plated through holes
5. All layers to be 1 oz copper.
6. All hole dimensions apply after plating
7. <= +/-0.003 inches.
8. All plated through holes to have a minimum copper thickness
9. 0.001 inch.
10. Layer to Layer registration shall be
11. <=0.003 inches of true position.
12. Apply solder mask over bare copper both sides.
13. Solder mask to be LPI
14. Color transparent green.
15. Exposed conductive surfaces to be covered by (ROHS compliant) Immersion.
16. Gold (or)
17. Silver (or)
18. ENEPIG
19. Silkscreen component & solder side using
20. White epoxy.
21. Ensure epoxy does not bleed to exposed metal.
22. Silkscreen on both sides if comports are on both sides.
23. Fabricate in accordance with IPC-6011 & 6012, Class 2.
24. Board shall meet requirements of UL796 with a flammability of 94V-0.
25. UL Logo shall be located on PCB.
26. Outline dimensions are
27. +/-0.010 unless otherwise noted.
28. Remove all burrs and sharp edges.
29. Mechanical Layer 2 of Gerber is PCB Outline.
30. Layer Stack order:

.GTL

.G1

.G2

.GBL

Notes: Contact Jim Blesener 763-398-0456 x2102 ([jimb@powerengmfg.com](mailto:jimb@powerengmfg.com)) with questions.