B2829 PCB SPECIFICATIONS

1. Board Layers: 6
2. Layer Stack Order:
   ARTWORK_1: Top Component Layer (Signal_1, PAD_1), 1oz.
   ARTWORK_2: POWER PLANE (AGND), 1oz.
   ARTWORK_3: POWER PLANE (VCC), 1oz.
   ARTWORK_4: POWER PLANE (VEE), 1oz.
   ARTWORK_5: POWER PLANE (GROUND), 1oz.
   ARTWORK_6: Bottom Side Layer (Signal_2, PAD_2), 1oz.
3. Apply silkscreen on both side:
   Artwork_7: Top component side silkscreen.
   Artwork_8: Bottom component side silkscreen.
4. Apply solder mask over bare copper on both side:
   Artwork_9: Top component side solder mask.
   Artwork_10: Bottom component side solder mask.
5. Solder paste photo plots for assembly stencils:
   Artwork_11: Top solder paste mask
   Artwork_12: Bottom csolder paste mask
6. Material: FR4 with Tg >170C.
7. Board thickness: 0.062 +/- 0.010.
8. All layers use 1 oz copper before plating.
9. Differential trace impedance control at 100 ohms +/- 10%. Trace/gap/trace = 5/5/5 mils
10. Minimum trace/gap = 5/5 mils
11. Ni/Au finish over bare copper.
12. Apply solder mask over bare copper
13. Send back photo plots and layer stack parameters for rechecking.
14. All dimensions are in inches unless otherwise noted.
15. Milling data file is provided for board cutting.
16: Contact person:
   Fukun Tang/Electronics Engineer
   Electronics Development Group
   University of Chicago
   Tel: (773)-834-4286, Fax: (773)-702-2971
   University of Chicago
   ELECTRONICS DEVELOPMENT GROUP
   TITLE
   B2829 Specification Drawing
   SHEET 1 OF 1
   DATE OCT-1-2013
   DRAWN F. Tang
   REV 1.0