

Drawing notes:

1) Fabricate PCB in accordance with IPC-6012, Class 2; per IPC-6011.

2) Materials:

- 1) Laminate and Prepreg (B-stage) to be in accordance with IPC-4101/21.
- 2) Copper foil to be in accordance with IPC-MF-150. Unless otherwise specified, all copper weight for inner signal layers and inner plane layers to be 1/2 oz. Outer layers to be 1.0 oz. Copper weight is to be considered finished.

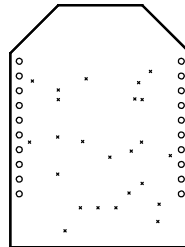
3) All holes shall be located within 0.1mm (0.004in) Diameter of True Position. Layer to layer registration shall be within 0.1mm (0.004in).

4) Finish:

- 1) All exposed conductive pattern areas not covered with solder mask or other plating shall be Hot Air Solder Leveled using eutectic Tin Lead Solder (Sn63/Pb37), per ANSI/J-STD-006 (HASL).
- 2) Apply liquid photo imageable solder mask (color green) per IPC-SM-840, class H, to both sides of the board over bare copper. Via holes covered with solder mask do not need to be plugged. Only solder mask images that are the same size as the component pads may be enlarged, and shall not be enlarged beyond 0.08mm (0.003in) per side or 0.15mm (0.006in) overall. All other solder mask images shall not be enlarged.
- 3) Silkscreen shall be white, permanent, organic, non-conductive ink. There shall be no silkscreen on any solderable component pad.

5) Marking:

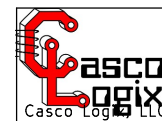
- 1) Board part number and revision letter is rendered in silkscreen as shown in the gerber files. Revision letter should be identical to this drawing.



Drill Map:

- 0.30mm / 0.012" (24 holes)
- 0.80mm / 0.031" (20 holes)

	SILKSCREEN, TOP
	SOLDERMASK, TOP
	1.0 OZ COPPER, TOP
	PREPREG
	0.5 OZ COPPER, LAYER 1
	CORE
	0.5 OZ COPPER, LAYER 2
	PREPREG
	1.0 OZ COPPER, BOTTOM
	SOLDERMASK, BOTTOM
	SILKSCREEN, BOTTOM



Sheet:

File: DorjiBee.kicad_pcb

Title: Dorji DBM01 Bluetooth Low Energy XBee Module

Size: USLetter Date: 2016-11-24

Rev: 1.0

KiCad E.D.A. kicad 4.0.4-stable

Id: 1/1