

Required files:
See Readme

Specifications:

All units are in inches unless otherwise specified.

1. Materials

Beginning foil thickness 1/2oz min. but shall meet the minimum requirements below:

- Layer 1 - Top Layer, 1 oz copper, 0.0014
- Dielectric - Rogers 4003, 0.016 +/- 0.0006
- Layer 2 - Ground Plane, 1 oz copper, 0.0014
- Dielectric - Fr4, (adjust this layer to meet thickness target)
- Layer 3 - Ground Plane, 1 oz copper, 0.0014
- Dielectric - Rogers 4003, 0.016 +/- 0.0006
- Layer 4 - Bottom Layer, 1 oz copper, 0.0014

Total Board Thickness = 0.062 +/- 0.005 including top and bottom soldermask.

2. Hole Plating

Copper plate 0.0010/0.0015 thick in Thru Hole Vias (Approx. 0.0015 on surface).

3. Finish

ENIG

Thickness 3-8 micro inches.

Solder Mask

Liquid Photo Imagable (LPI) Solder Mask Over Bare Copper (SMOBC) on Both Sides of PCB.

Color: Red or Approved Equal.

Silkscreen: White Non-Conductive Epoxy Ink.

Supplier ID and Lot Number shall be noted on bottom silkscreen.

4. Tolerances

Overall board dimension tolerance +/- 0.010.

Hole Diameter >= 0.100 +/- 0.005.

Hole Diameter < 0.100 +/- 0.003.

Layer to layer registration should be +/- 0.001.

All holes shall be located within hole to pad tangency.

5. Traces

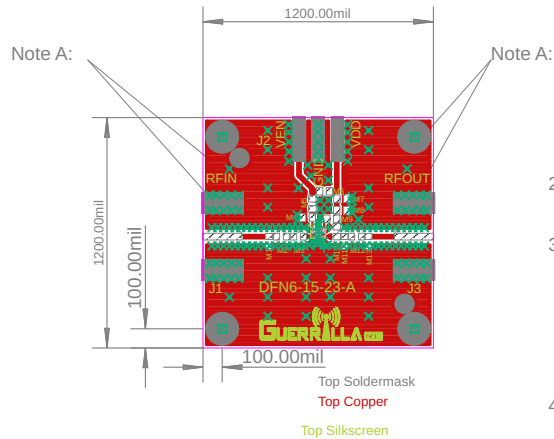
Conductors indicated as Note A shall be 0.030 +/- 0.001 wide and have a coplanar ground spacing of 0.020 +/- 0.001 Manufacturer shall test for lines indicated to be 50 Ohm +/- 5% and may adjust the width as needed.

Minimum conductor width is 0.007 +/- 0.001.

Minimum conductor to conductor spacing is 0.006.

TDR coupon required to be delivered with boards.

The controlled impedance on layer 1 shall be 50 Ohms +/- 5%.



Sym	Nº	Mils	MM	Qty	Plated
+	1	8	0.20	13	YES
x	2	10	0.25	166	YES
□	3	125	3.18	4	YES

REV	Description	Date
1	Initial release	09/08/2017



Dimensions are in inches unless otherwise specified.		SIZE		DOCUMENT NO. DFN6-15-23	REV A
Drawn By: Eugenio M. Proenza		SCALE NONE	Date: 09/08/2017	Sheet 1 of 1	