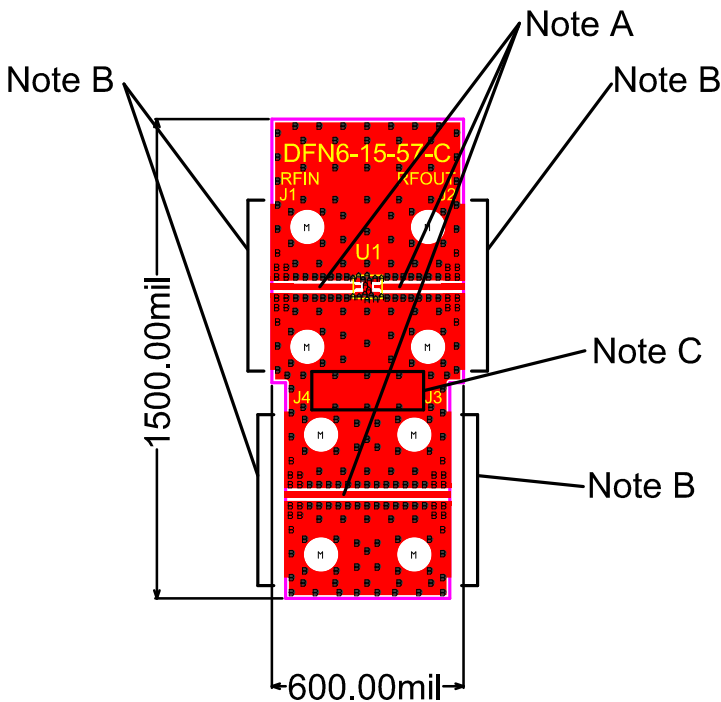


REV	Description	Date

- Specifications:
All units are in mils 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, 1.4mil
 - Dielectric - Rogers 4003, 12mil +/- 1mil
 - Layer 2 - Ground Plane, 1 oz copper, 1.4mil
 - Dielectric - Fr4 with 180Tg, (adjust this layer to meet thickness target)
 - Layer 3 - Ground Plane, 1 oz copper, 1.4mil
 - Dielectric - Rogers 4003, 12mil +/- 1mil
 - Layer 4 - Bottom Layer, 1 oz copper, 1.4mil
 - Total Board Thickness = 62mil +/- 5mil not including top and bottom solder mask.
 - 2. Hole Plating
 - Copper plate in Thru Hole Vias - 1.0mil thick minimum (Approx. 1.5mil 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.
 - Silkscreen is not to be applied to bottom of PCB
 - 4. Tolerances
 - Overall board dimension tolerance +/- 5mil.
 - Hole Diameter >= 100mil +/- 5mil
 - Hole Diameter < 100mil +/- 3mil
 - Layer to layer registration should be +/- 3mil.
 - All holes shall be located within hole to pad tangency.
 - 5. Traces
 - Conductors indicated as Note A shall be 20mil +/- 1mil wide and have a coplanar ground spacing of 10mil +/- 1mil Manufacturer shall test for lines indicated to be 50 Ohm +/- 5% and may adjust the width as needed.
 - Minimum conductor width is 6mil +/- 1mil.
 - Minimum conductor to conductor spacing is 6mil.
 - TDR coupon required to be delivered with boards.
 - The controlled impedance on layer 1 shall be 50 Ohms +/- 5%.
 - 6. RF Connections
 - Areas indicated as Note B shall have metal extending to the board edge and exposed when viewed from the side.
 - This note will apply to all layers in the indicated area.
 - Burrs and potential shorts will be accepted.
 - 7. UL Mark and Date code may be added to top layer silk screen in area noted in Note C. Lettering should be oriented on same axis as Guerrilla RF Logo lettering.



Top Overlay (Silk Screen)

Top Layer (Metal)

Drill Drawing

Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair
A	16	8.00mil (0.203mm)	PTH	Round	Top Layer - Bottom Layer
B	241	10.00mil (0.254mm)	PTH	Round	Top Layer - Bottom Layer
M	8	86.61mil (2.200mm)	NPTH	Round	Top Layer - Bottom Layer
	265 Total				

		Guerrilla RF		
Dimensions are in inches unless otherwise specified.	SIZE		DOCUMENT NO. DFN6-15-57-C	REV A
	SCALE NONE		Date: 4/20/2026	Sheet 1 of 1
Drawn By: R Williamson				