Date: 2022-12-30

FCC ID: R68OQ845US

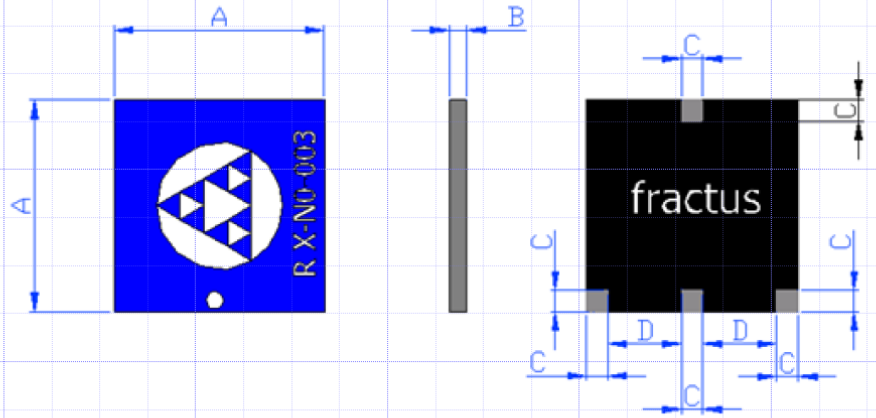
Declaration for KDB 996369 D02 Module Q&A

To whom it may concern,

Question 1. Layout of trace design, parts, antenna, connectors, and isolation requirements.

Description:

4.1. DIMENSIONS AND TOLERANCES



The white dot located on the top side of the antenna indicates the feed pad.

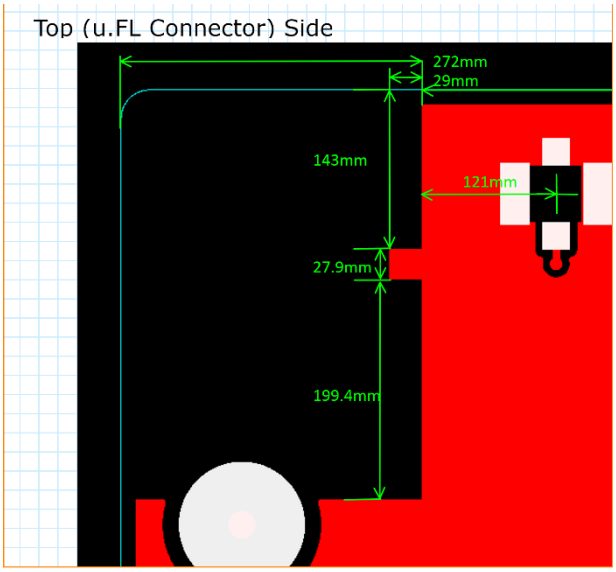
Measure	mm	Measure	mm
A	10.0 ± 0.2	C	1.0 ± 0.1
B	0.9 ± 0.2	D	3.5 ± 0.2

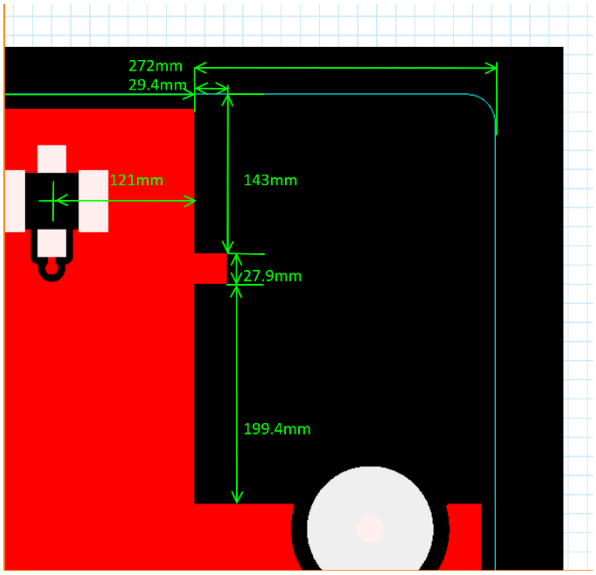
Figure 3 – Antenna Dimensions and Tolerances.

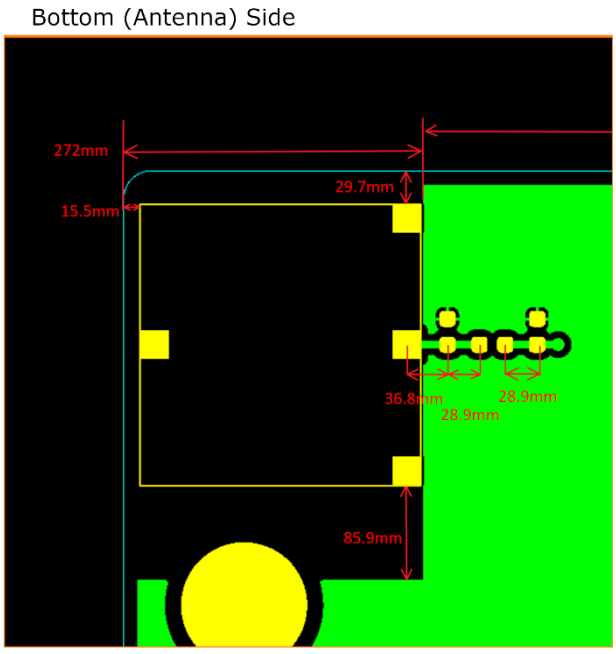
Technical Features	802.11 b/g	802.11a
Frequency Range	2.4 – 2.5 GHz	4.9 – 5.875 GHz
Peak Gain	3.9 dBi	6.0 dBi
VSWR	< 2:1	< 2:1
Average Efficiency	75.0 %	74.6 %
Isolation	> 12.0 dB	> 16.0 dB
Polarization	Linear	
Radiation Pattern	Omnidirectional	
Weight (approx.)	0.2 g	
Temperature	-40° C to +85° C	
Impedance	50 Ω	

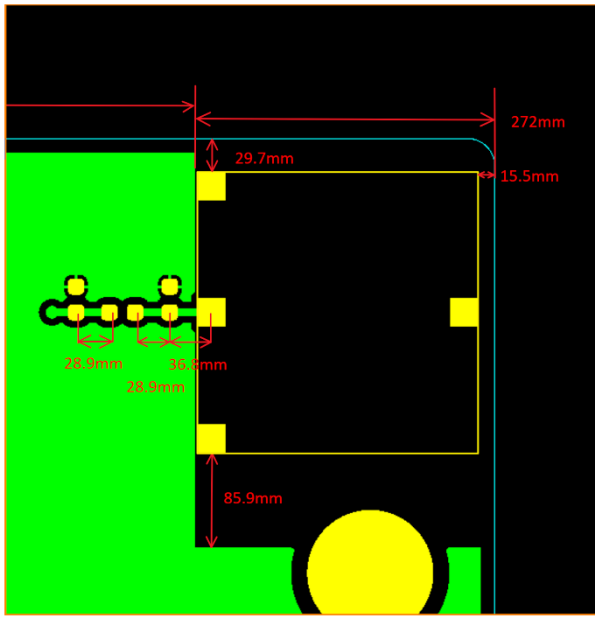
Question 2. Boundary limits of size, thickness, length, width, shape(s), dielectric constant, and impedance must be clearly described for each type of antenna.

Description:

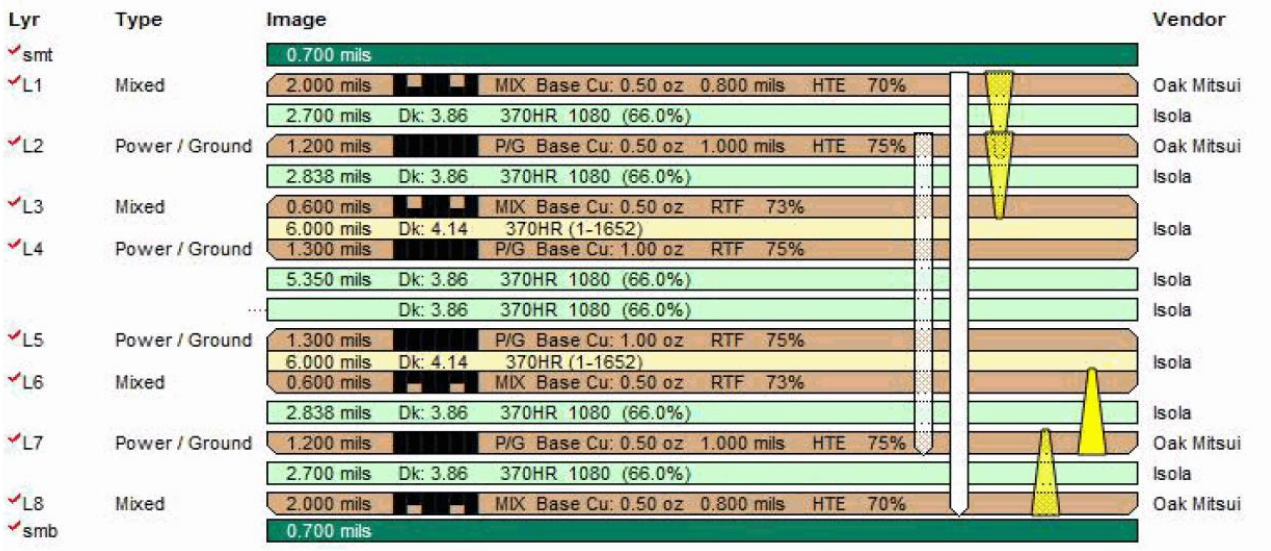








Stack up:



Dielectric Constant: See stack-up above (Dk)

Impedance: 50Ω

Question 3. Different antenna length and shapes affect radiated emissions, and each design shall be considered a different type; e.g., antenna length in multiple(s) of frequency wavelength and antenna shape (traces in phase) can affect antenna gain and must be considered.

Description:

There is only one type of antenna, no different type.

Question 4. The above data is to be provided by a Gerber file (or equivalent) for PC layout.

Description:

Please see information in Q.2

Question 5. Appropriate parts by manufacturer and specifications.

Description:

Manufacturer: Fractus Antennas

Part Number: FR05-S1-NO-1-003

Please refer to “Fractus_UM_FR05_S1_NO_1_003.pdf” for further information.

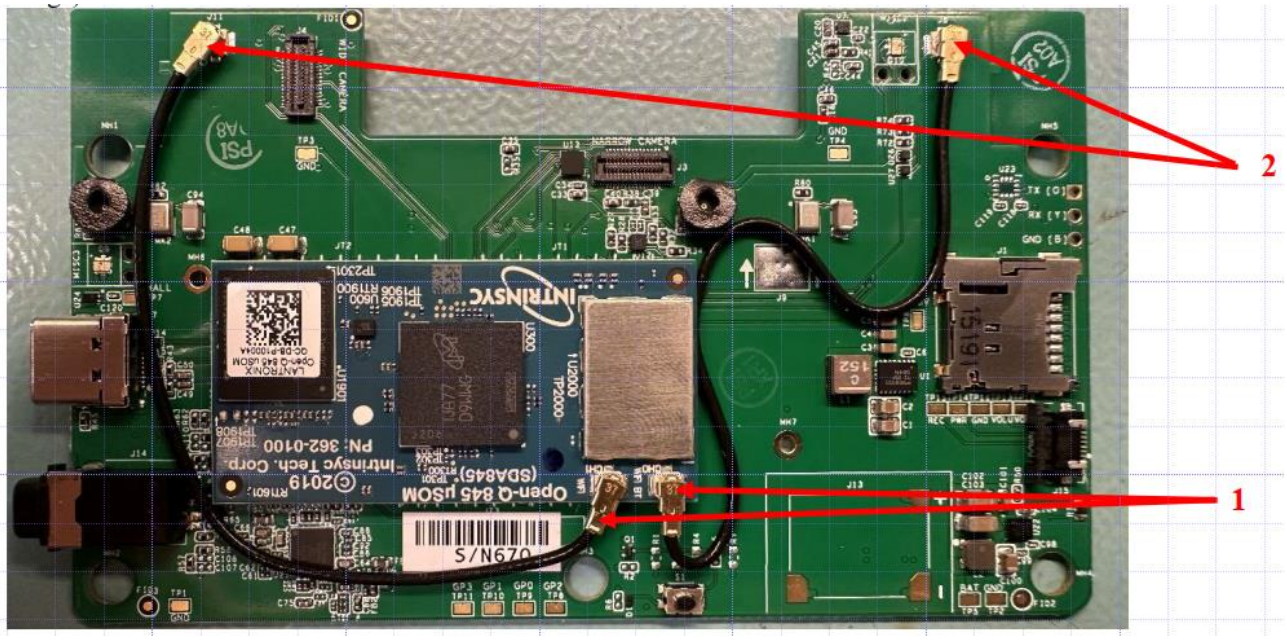
Question 6. Test procedures for design verification.

Description:

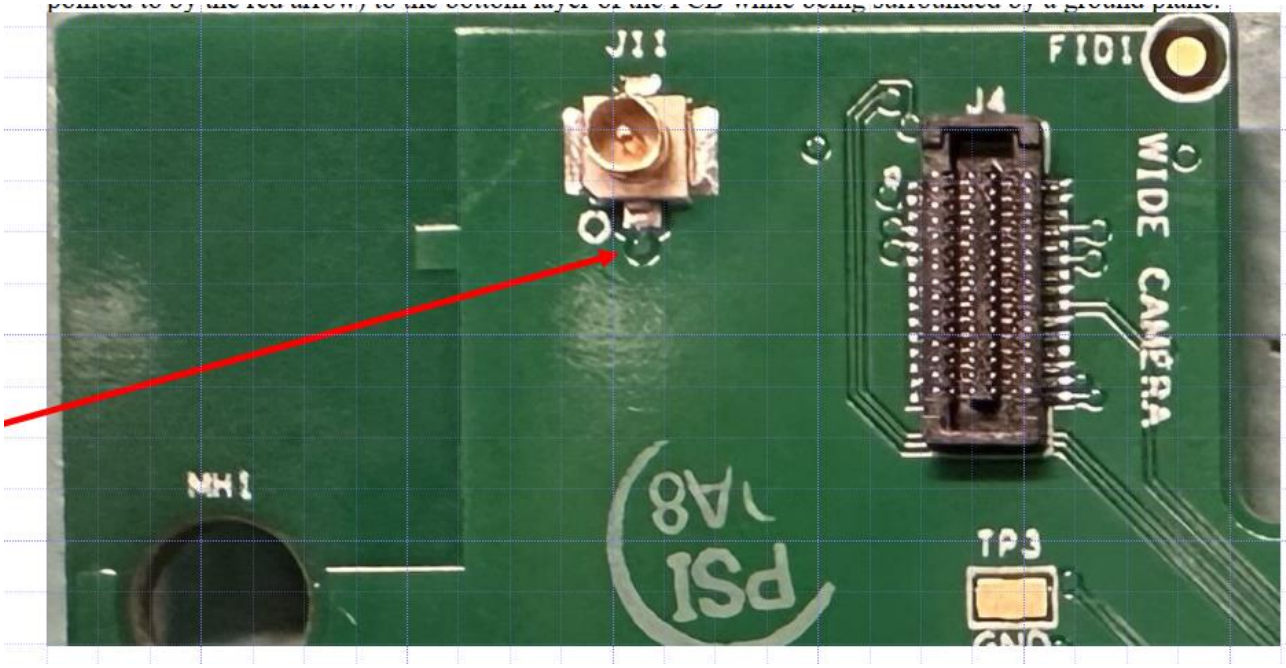
Designer should check their antenna routing using the following guidelines.

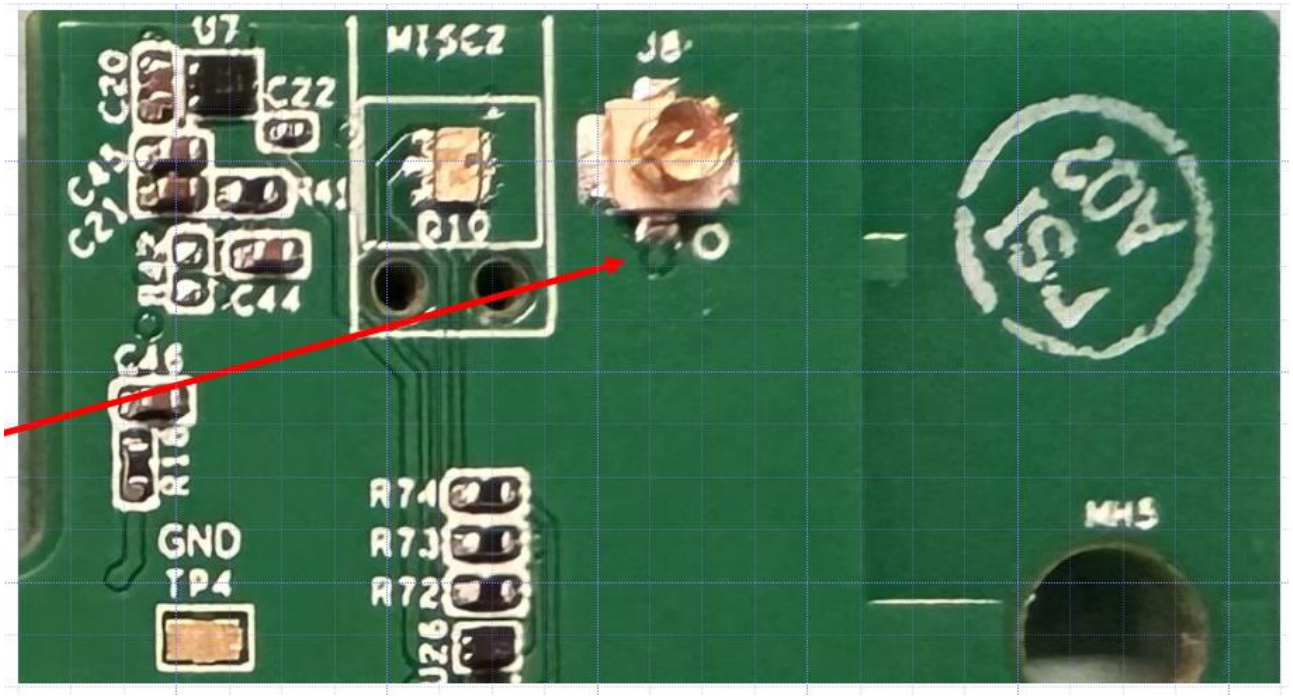
Top Layer:

In the image below, the antenna paths are being routed from the Lantronix QC-DB-P10004A SOM u.FL connectors (marked as 1 in the image), to u.FL connectors on the PCB board (marked as 2 in the image) via u.FL cables. The cable lengths are 7.9 cm & 10.2 cm.



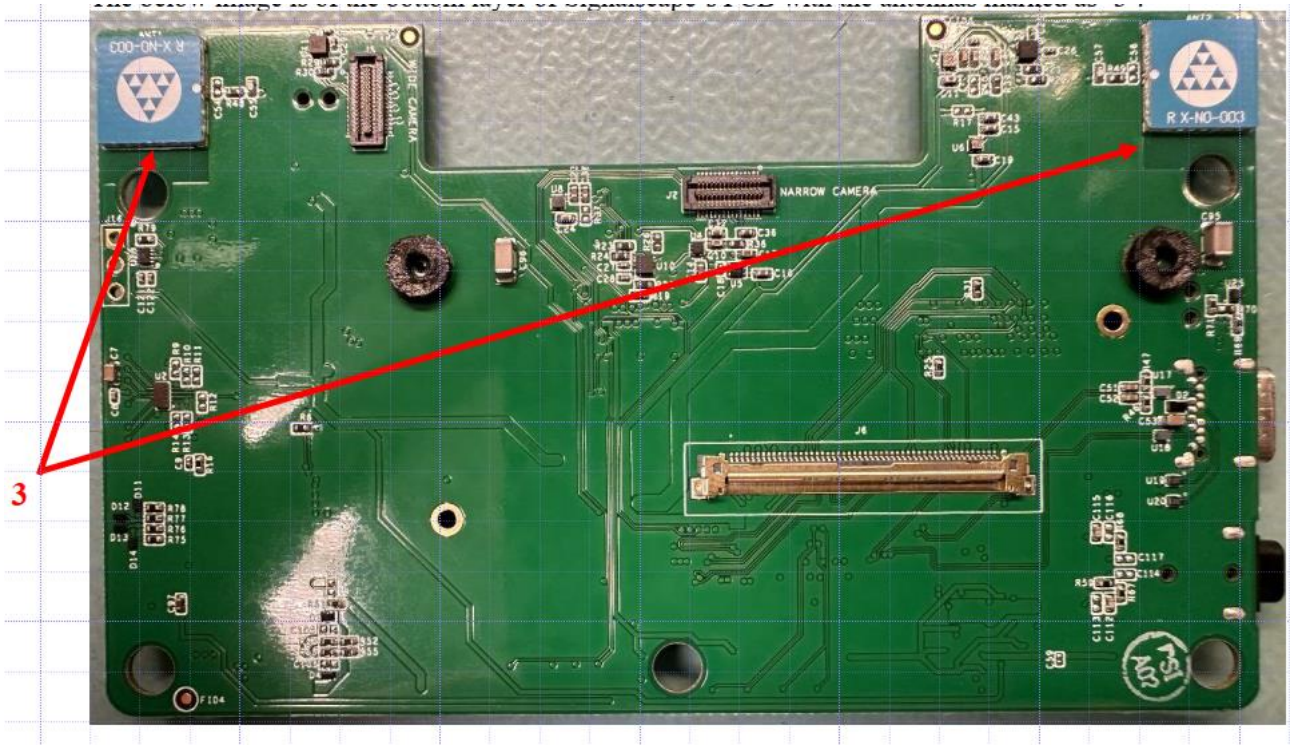
The next two images are close ups of the regions marked as ‘2’ in the above image. The intention is to show how the copper trace comes out of the u.FL connector and immediately vias down (see circle being pointed to by the red arrow) to the bottom layer of the PCB while being surrounded by a ground plane.



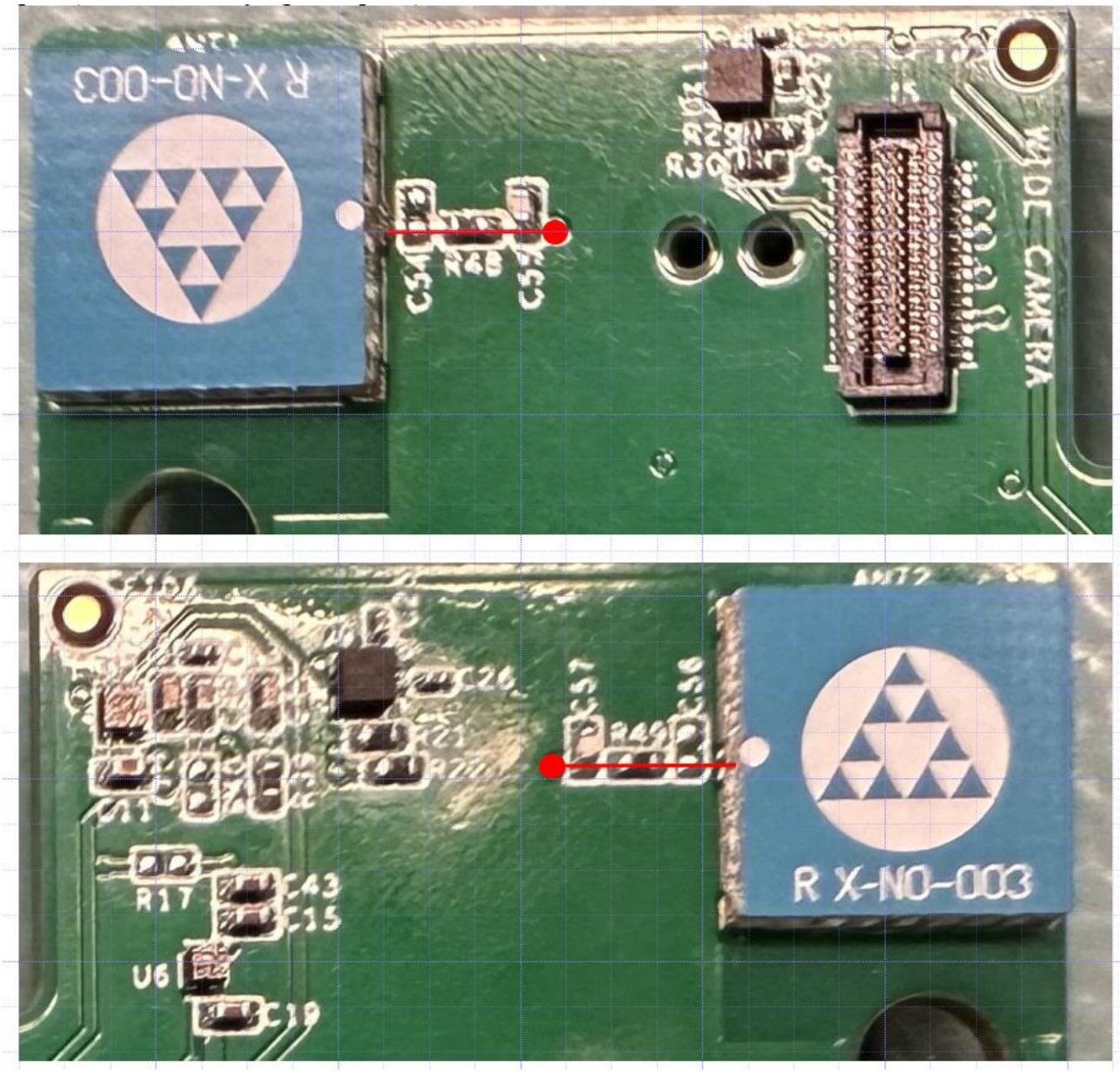


Bottom Layer:

The below image is on the bottom layer of the PCB with the antennas marked as ‘3’.



Below image is a close-up of the antenna regions showing the via from the top layer (denoted by red circle) and the copper trace (denoted by red line) that routes through a tuning circuit to the antenna pads (all surrounded by a ground plane).



Question 7. Production test procedures for ensuring compliance.

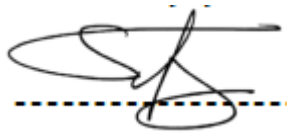
Description:

Customer should specify all RF trace impedance as 50Ω (+/- 5Ω) using controlled impedance PCB fabrication.

After assembly, customer should measure TX power, sensitivity, and any unwanted signal from surrounding circuit couple into the radio path using a spectrum analyzer. If unwanted coupling detected, distance between RF path and surrounding circuit may needs to be re-routed.

Thank you for your attention.

Sincerely yours,



Fathi Hakam / VP of Engineering
Tel: +1 (949) 453-3990
Fax: +1 (949) 453-3995
E-mail: fhakam@lantronix.com