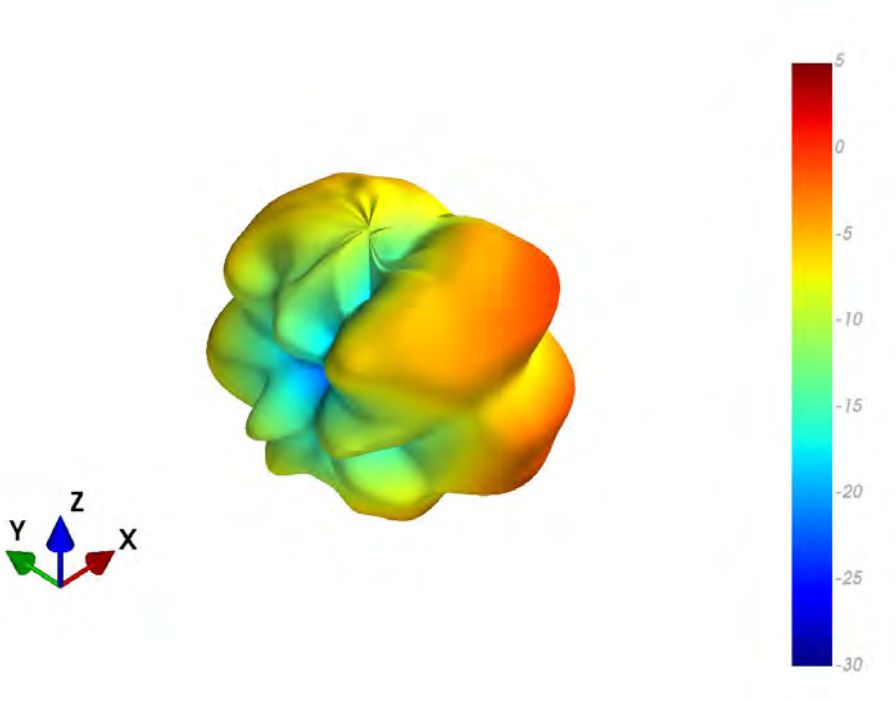
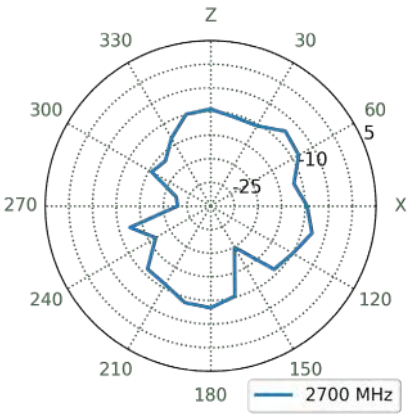
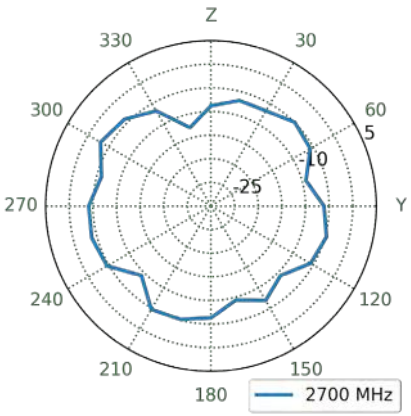
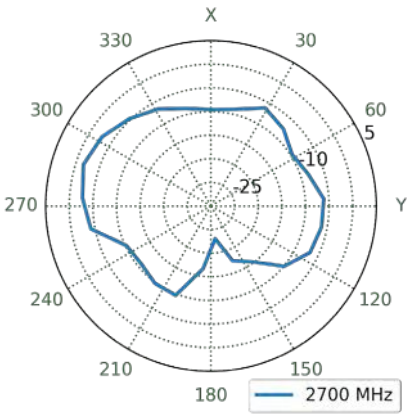
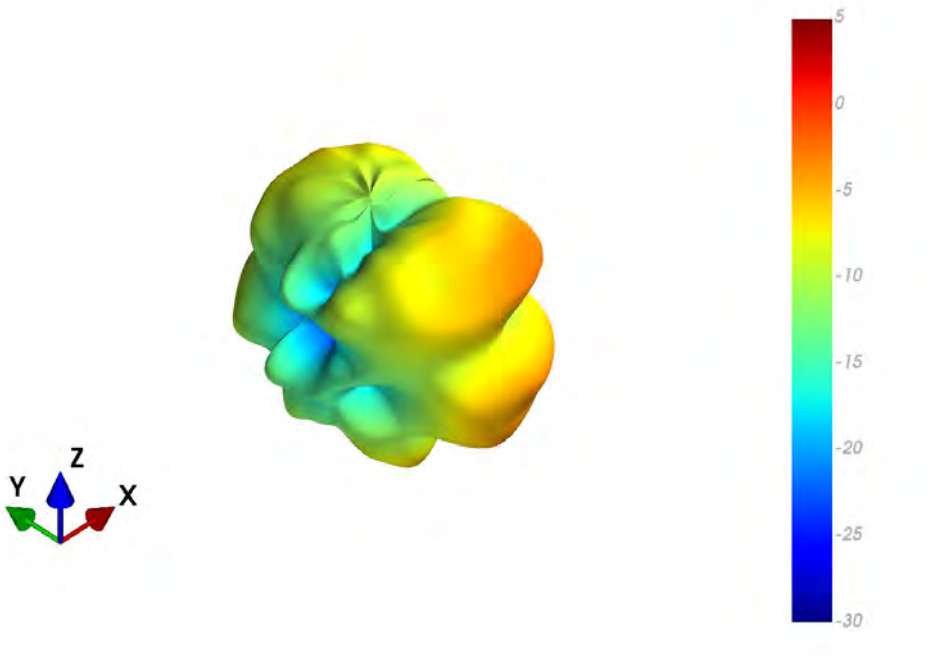
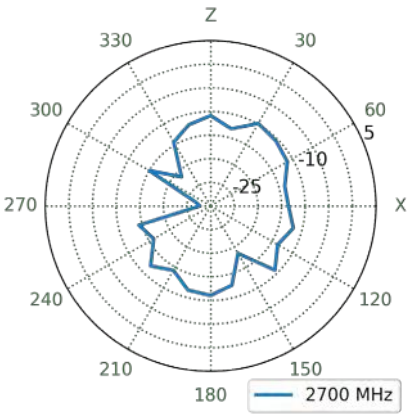
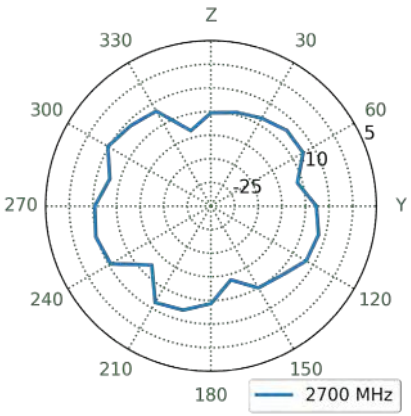
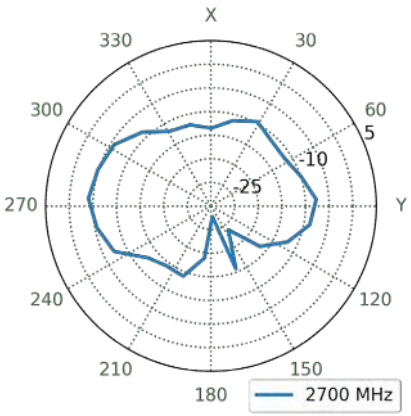
4.35 EMEA Patterns at 2700 MHz



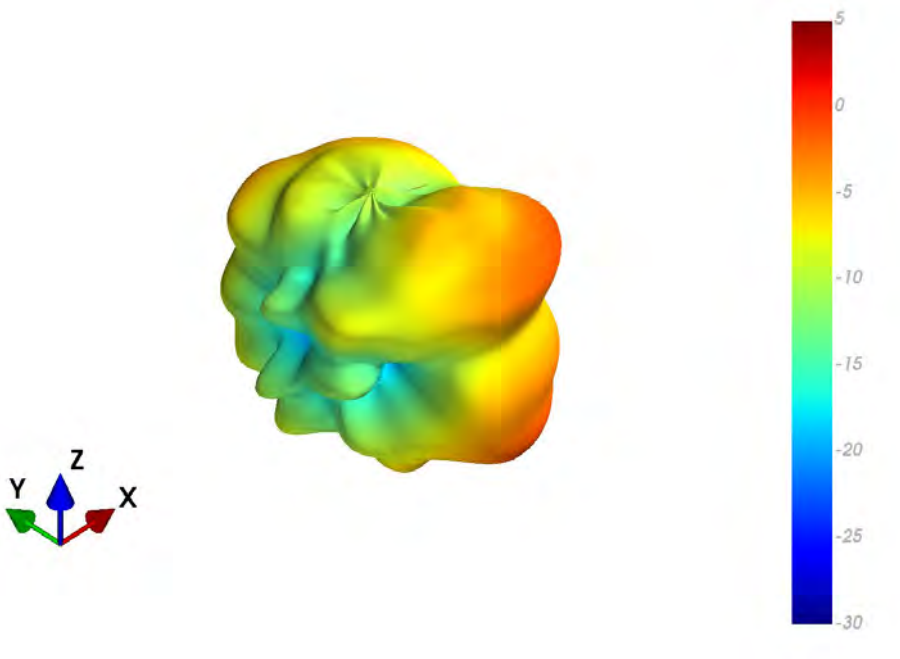
XZ Plane	YZ Plane	XY Plane
		

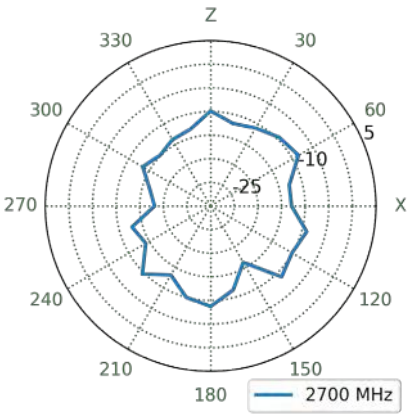
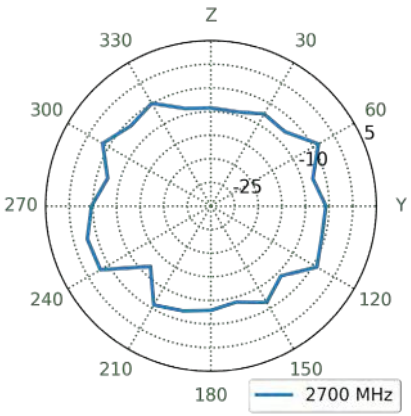
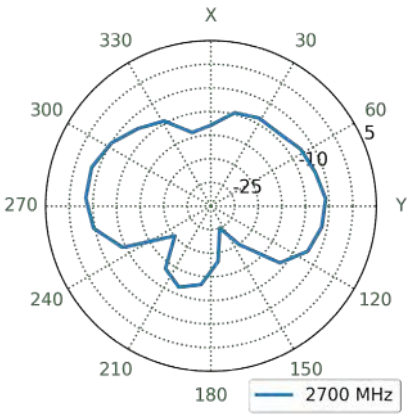
4.36 NA Patterns at 2700 MHz



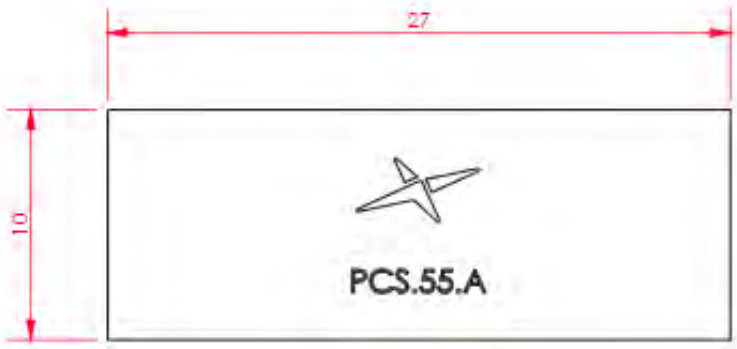
XZ Plane	YZ Plane	XY Plane
		

4.37 WW Patterns at 2700 MHz



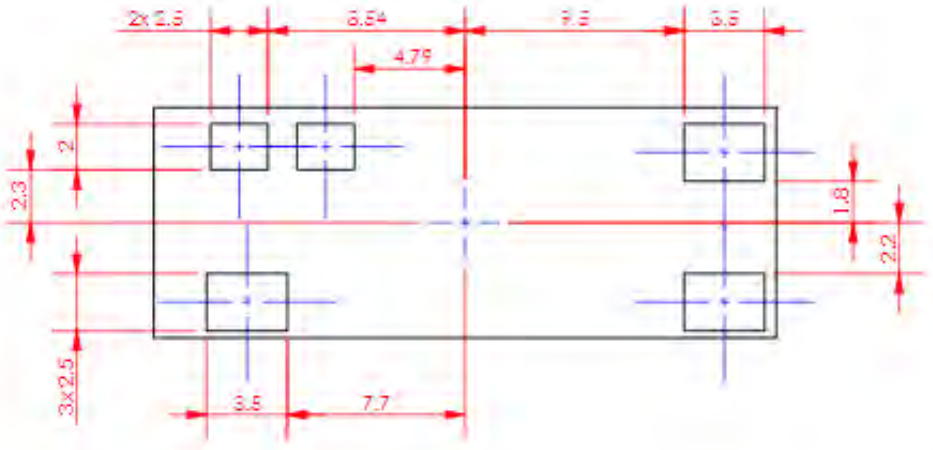
XZ Plane	YZ Plane	XY Plane
		

5. Mechanical Drawing



TOP VIEW

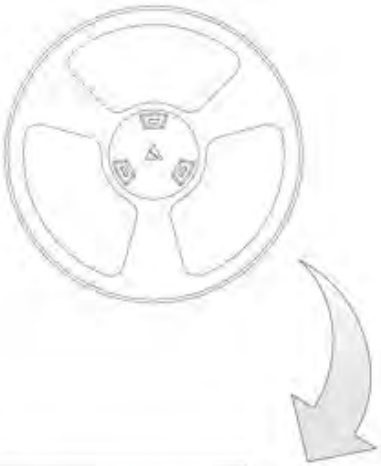
FRONT VIEW



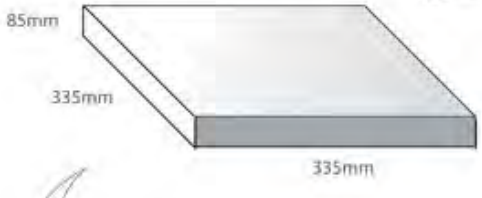
BOTTOM VIEW

6. Packaging

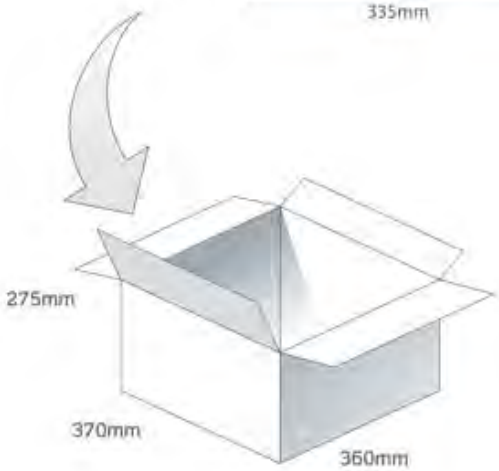
1000 pc PCS.55.A per reel
Dimensions – Ø330*60
Weight – 1630g

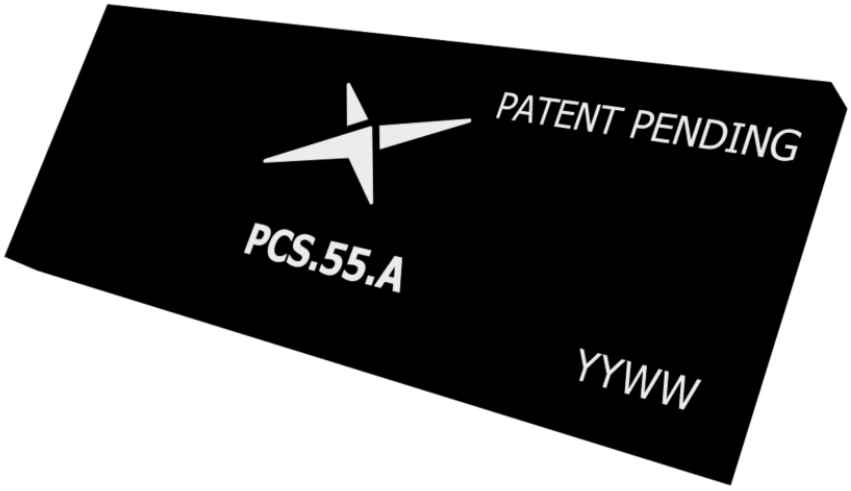


1000 pc PCS.55.A / 1 reel in small box
Dimensions – Ø335*335*85mm
Weight – 1.9Kg



3 reels, 3000 pcs in one carton
Carton dimensions – 370*360*275mm
Weight – 6.5Kg



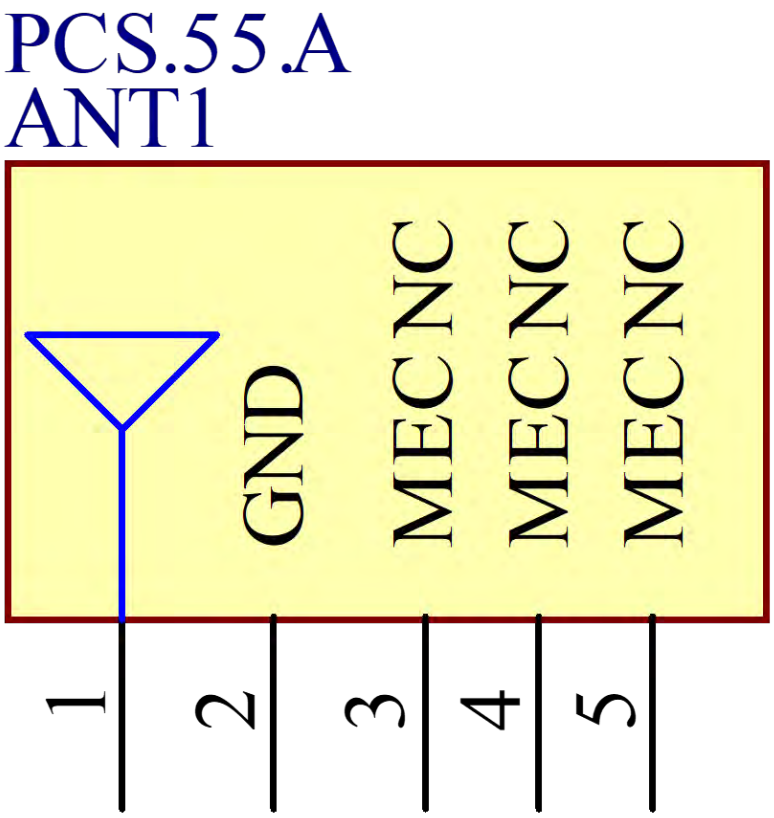

PATENT PENDING
PCS.55.A
YYWW


PATENT PENDING
PCS.55A
YYWW
ANT1
L1
C1
C2
TAOGLAS
PCSD.55.A Evaluation Board
D02 YYWW

7.1 Schematic and Symbol Definition

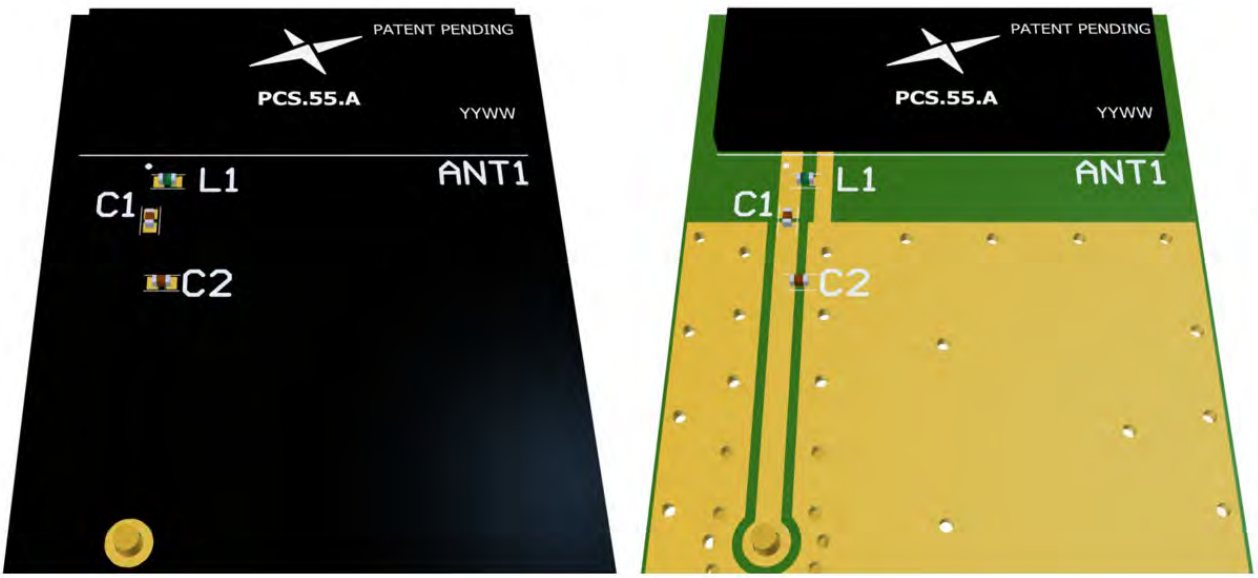
The circuit symbol for the antenna is shown below. The antenna has 5 pins with only two pins (Pin 1 and Pin 2) as functional. Pins 3, 4 and 5 are for mechanical strength.

Pin	Description
1	RF Feed
2	Ground
3, 4, 5	Mechanical, Not Connected



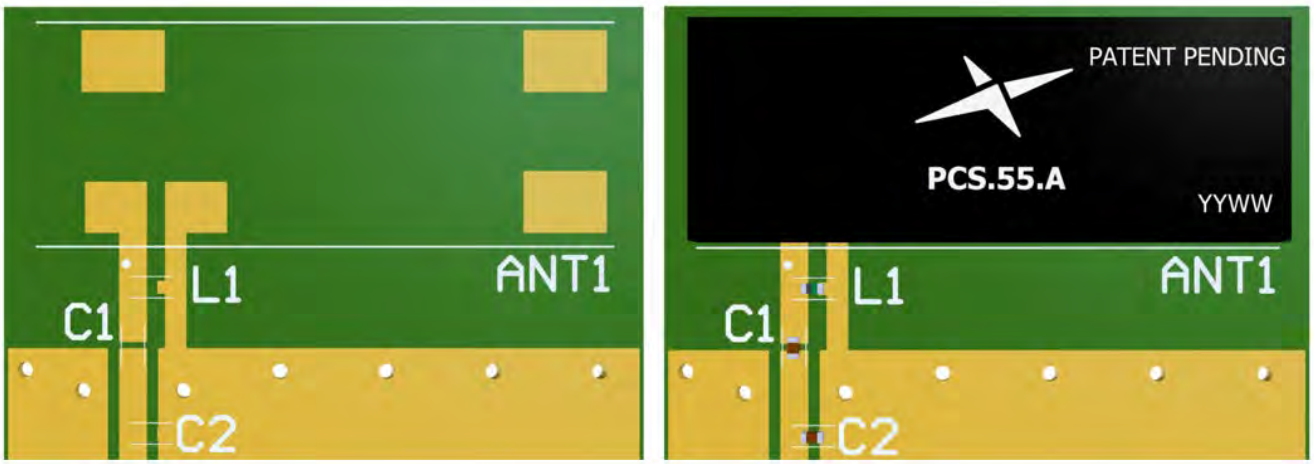
7.2 Antenna Integration

For any given PCB size, the antenna should ideally be placed on the PCB's shortest side, to take advantage of the ground plane. Optimized matching components can be placed as shown.



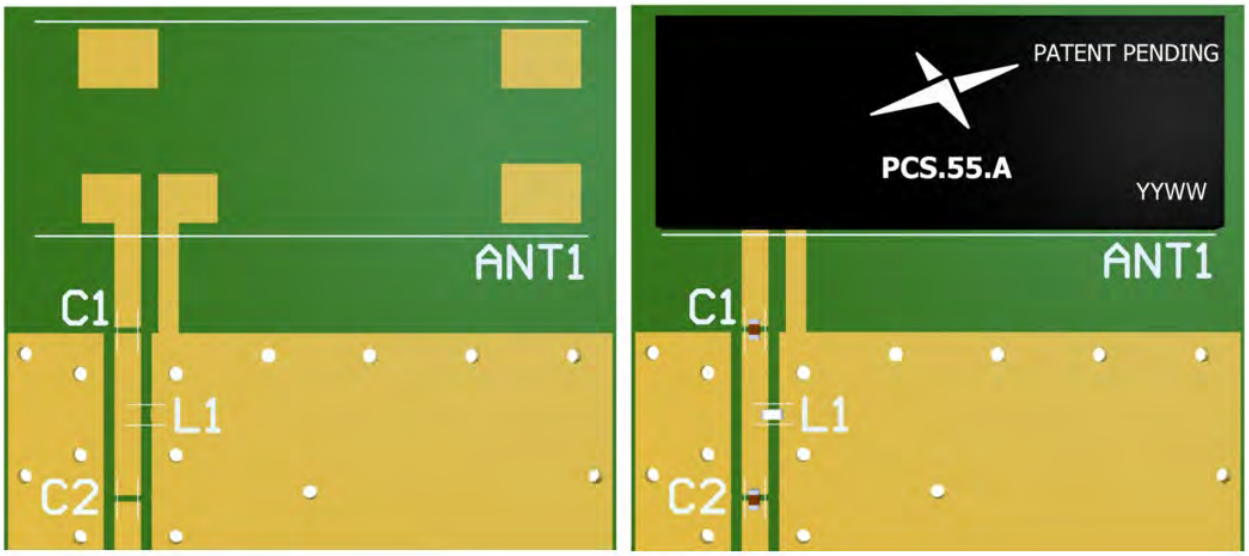
7.3 PCB Layout - NA

The footprint and clearance on the PCB must meet the layout drawing in section (7.12). Note the placement of the optimized components. L1 is placed as close as possible to the RF feed (pad 1) within the copper keep out area. C1 is then placed tightly in series with C2 placed in parallel after that.



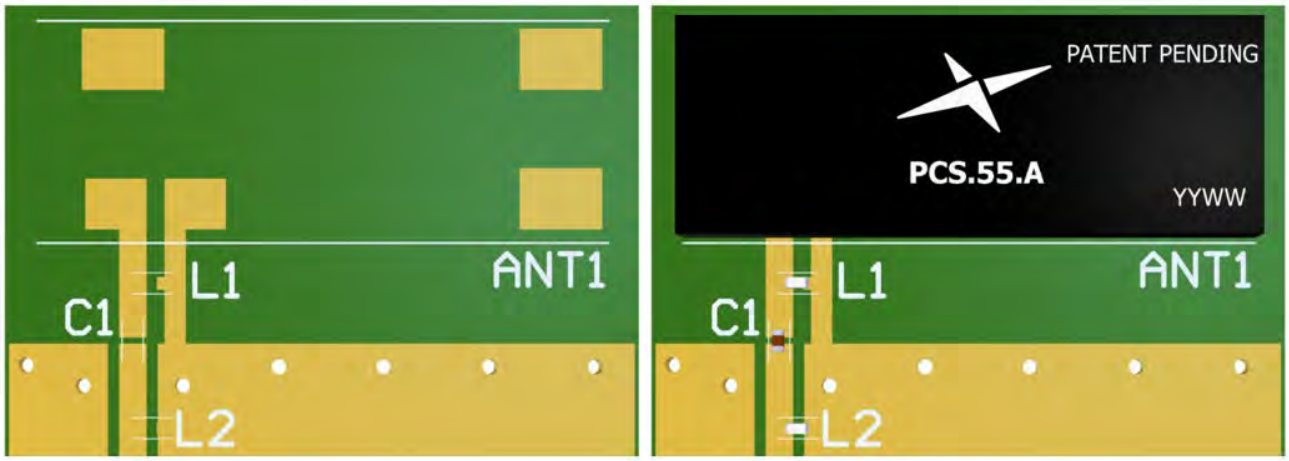
7.4 PCB Layout - EMEA

The footprint and clearance on the PCB must meet the layout drawing in section (7.12). Note the placement of the optimized components. C1 is placed as close as possible to the RF feed (pad 1) across the copper keep out area. L1 is then placed tightly in parallel with C2 placed in series after that.



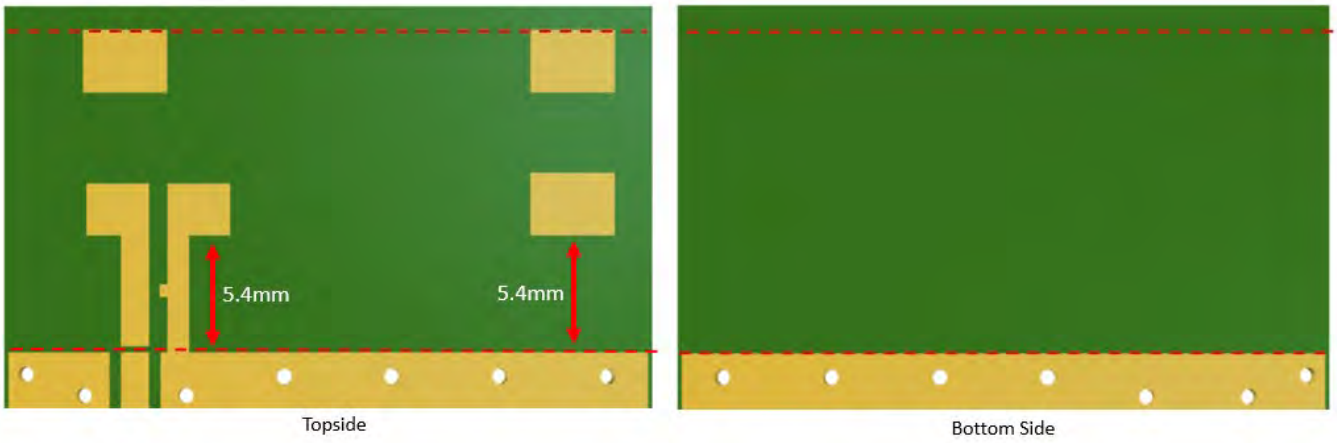
7.5 PCB Layout - WW

The footprint and clearance on the PCB must meet the layout drawing in section (7.12). Note the placement of the optimized components. L1 is placed as close as possible to the RF feed (pad 1) within the copper keep out area. C1 is then placed tightly in series with L2 placed in parallel after that.



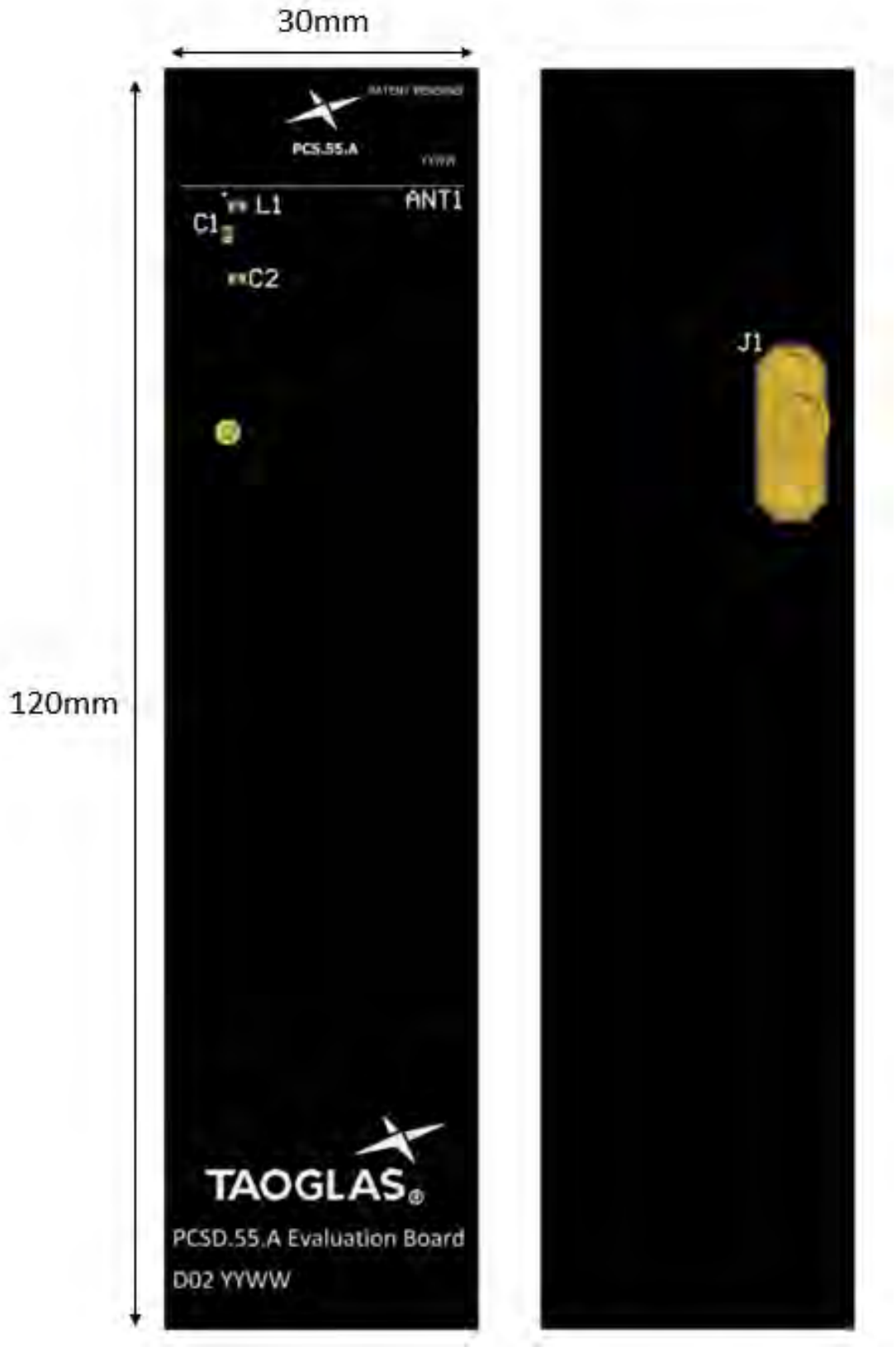
7.6 PCB Clearance

Below shows the antenna footprint and clearance through ALL layers on the PCB. Only the antenna pads and connections to feed and GND are present within this clearance area (marked RED). The clearance area extends to 5.4mm from the antenna pads to the ground area. This clearance area includes the bottom side and ALL internal layers on the PCB.



7.7 Evaluation Board

.



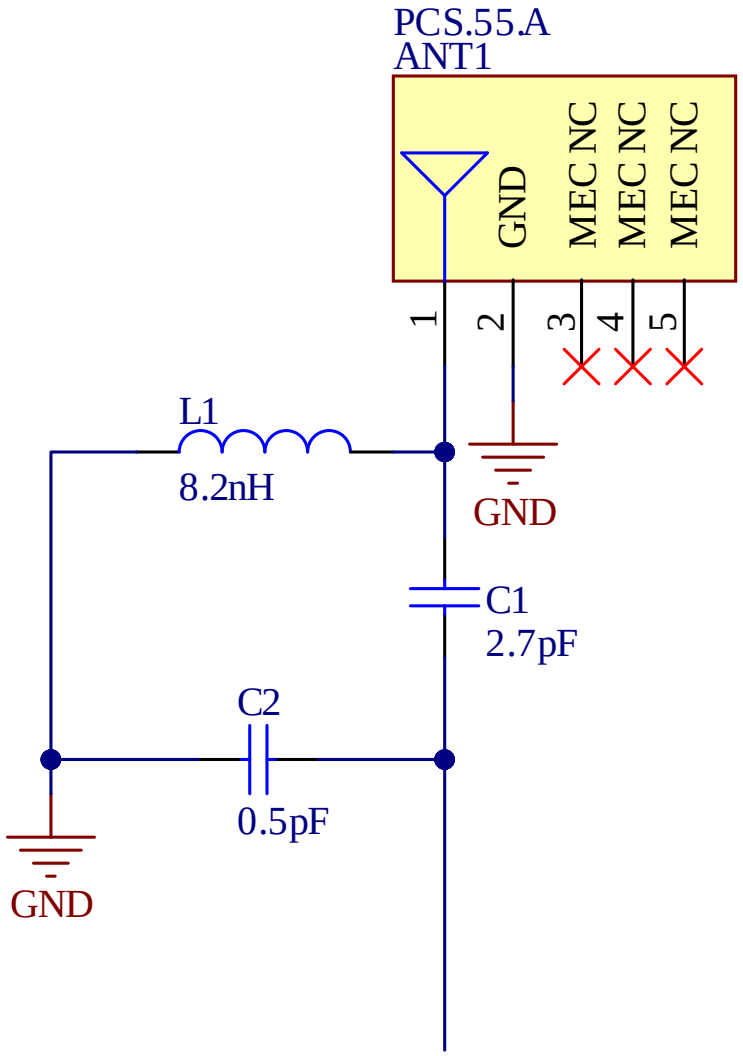
7.8 Evaluation Board Ground Plane Length

Ground Plane Length: 104mm

7.9 Matching Circuit - NA

Matching components with the PCS.55.A are recommended for the antenna to have optimal performance on the evaluation board, located in the spaces specified in the above images. Additional matching components may be necessary for your device, so we recommend incorporating extra component footprints, forming a “pi” network, between the cellular module and the edge of the ground plane.

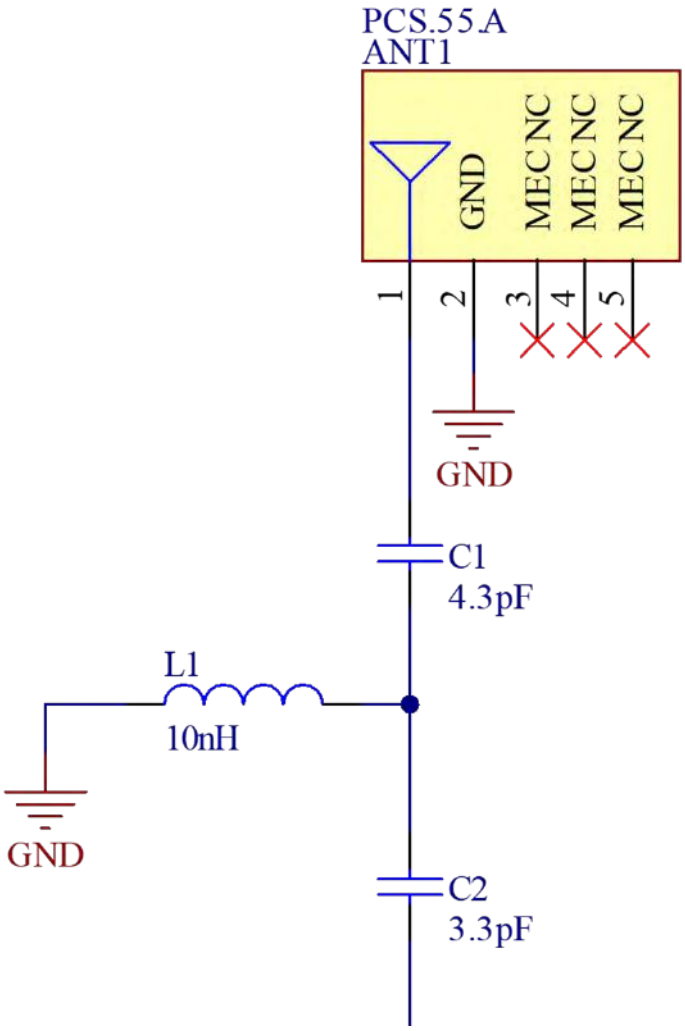
Designator	Type	Value	Manufacturer	Manufacturer Part Number
L1	Inductor	8.2nH	Murata	LQG15HS8N2G02D
C1	Capacitor	2.7pF	Murata	GCM1555C1H2R7BA16D
C2	Capacitor	0.5pF	Murata	GRM1555C1HR50CA01D



7.10 Matching Circuit - EMEA

Matching components with the PCS.55.A are recommended for the antenna to have optimal performance on the evaluation board, located in the spaces specified in the above images. Additional matching components may be necessary for your device, so we recommend incorporating extra component footprints, forming a “pi” network, between the cellular module and the edge of the ground plane.

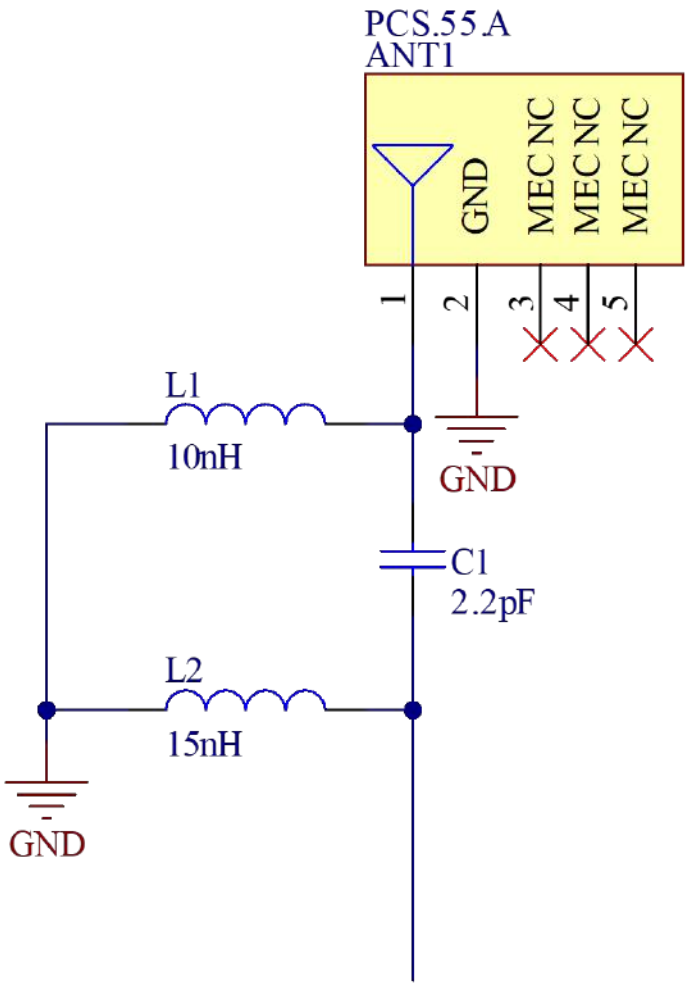
Designator	Type	Value	Manufacturer	Manufacturer Part Number
L1	Inductor	10nH	Murata	LQG15HS10NH02D
C1	Capacitor	4.3pF	Murata	GJM1555C1H4R3BB01D
C2	Capacitor	3.3pF	Murata	GJM1555C1H3R3BB01D



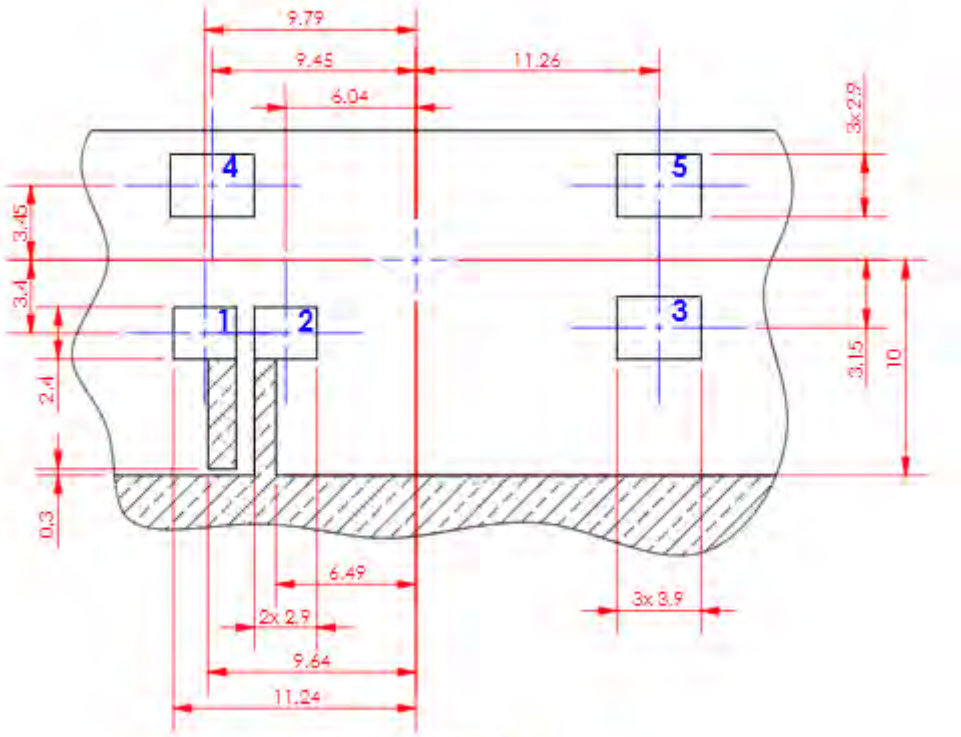
7.11 Matching Circuit - WW

Matching components with the PCS.55.A are recommended for the antenna to have optimal performance on the evaluation board, located in the spaces specified in the above images. Additional matching components may be necessary for your device, so we recommend incorporating extra component footprints, forming a “pi” network, between the cellular module and the edge of the ground plane.

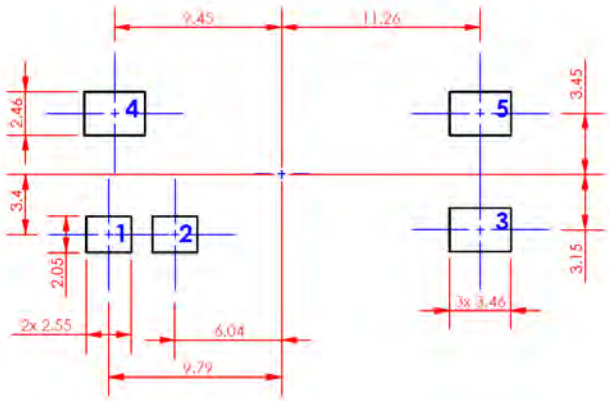
Designator	Type	Value	Manufacturer	Manufacturer Part Number
L1	Inductor	10nH	Murata	LQG15HS10NH02D
L2	Inductor	15nH	Murata	LQG15HN15NG02D
C1	Capacitor	2.2pF	Murata	GRM1555C1H2R2CA01D



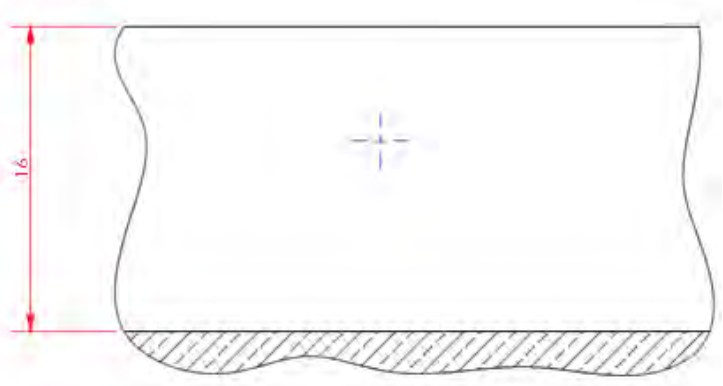
7.12 Footprint



FOOT PRINT PCB



FOOTPRINT SOLDER PASTE

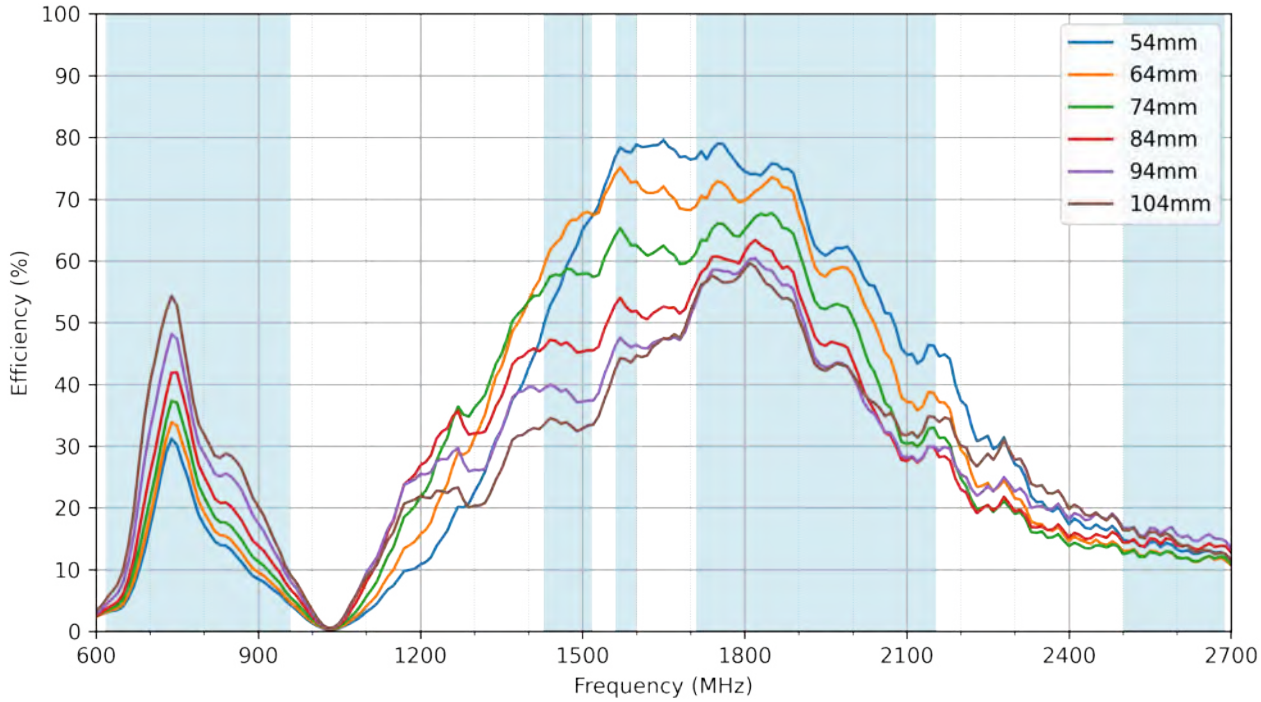


GROUND CLEARANCE BOTTOM VIEW

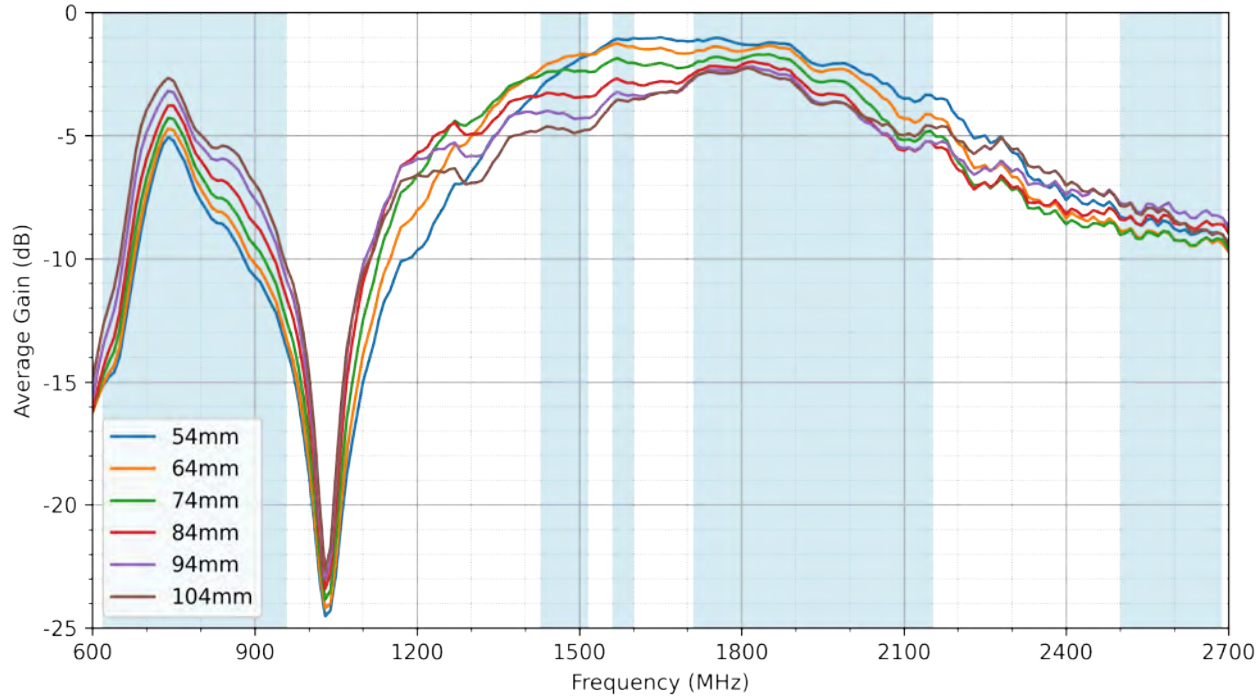
8. Application Note

The effect of shortening the ground plane on antenna performance was evaluated. Using the evaluation board of the PCS.55.A, the PCB was cut back 10mm at a time and tested in an anechoic chamber. The results for North America (NA), Europe, Middle East, Africa (EMEA), and Worldwide (WW) are shown here:

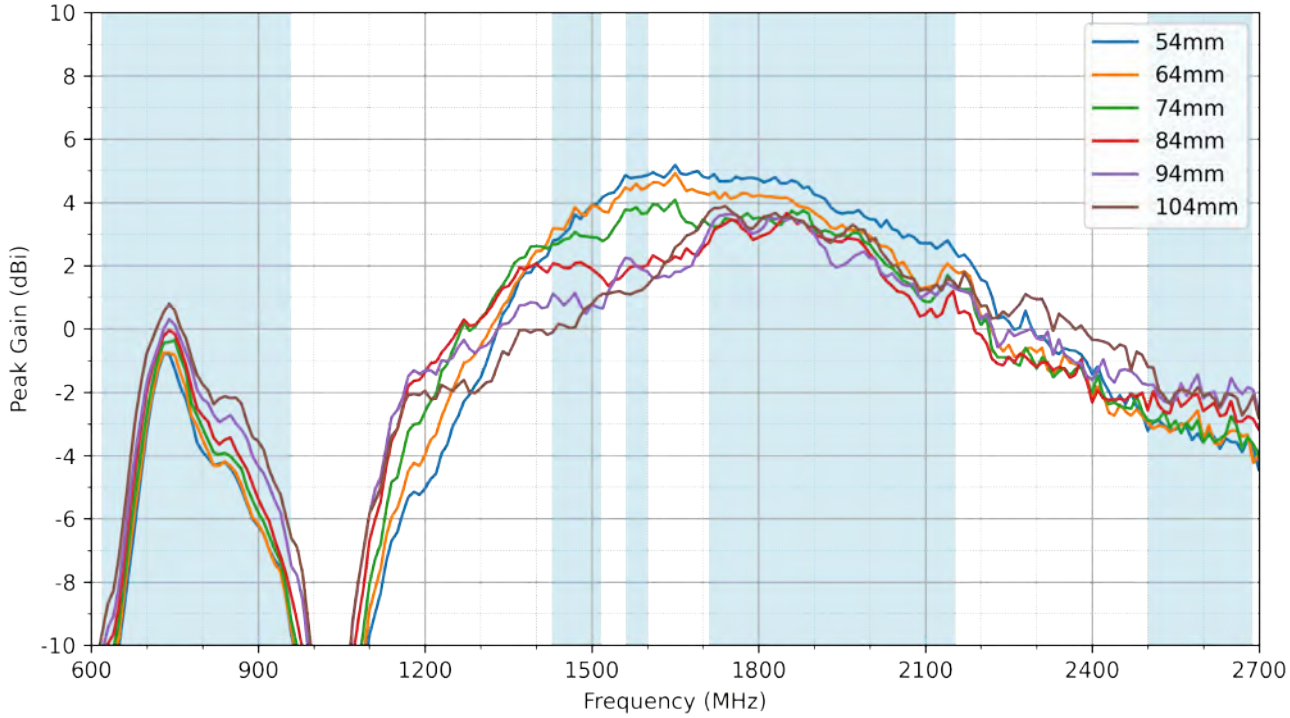
8.1 Efficiency – North America



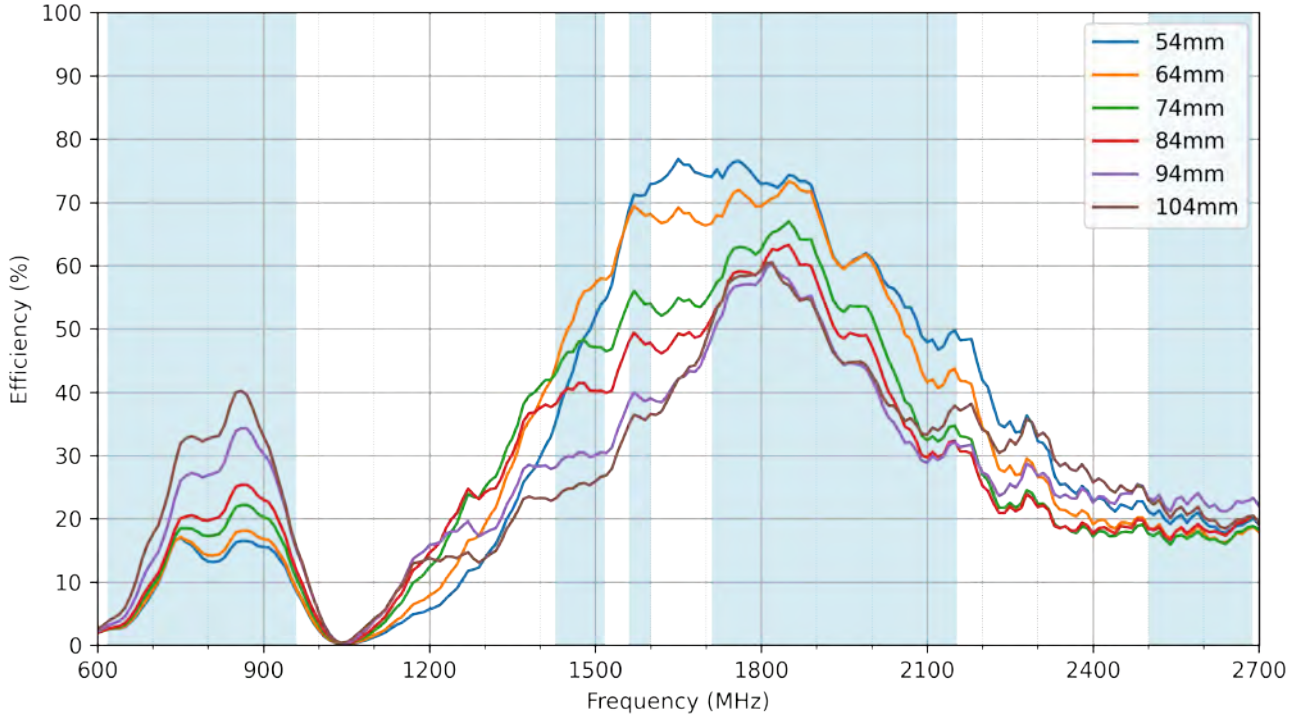
8.2 Average Gain – North America



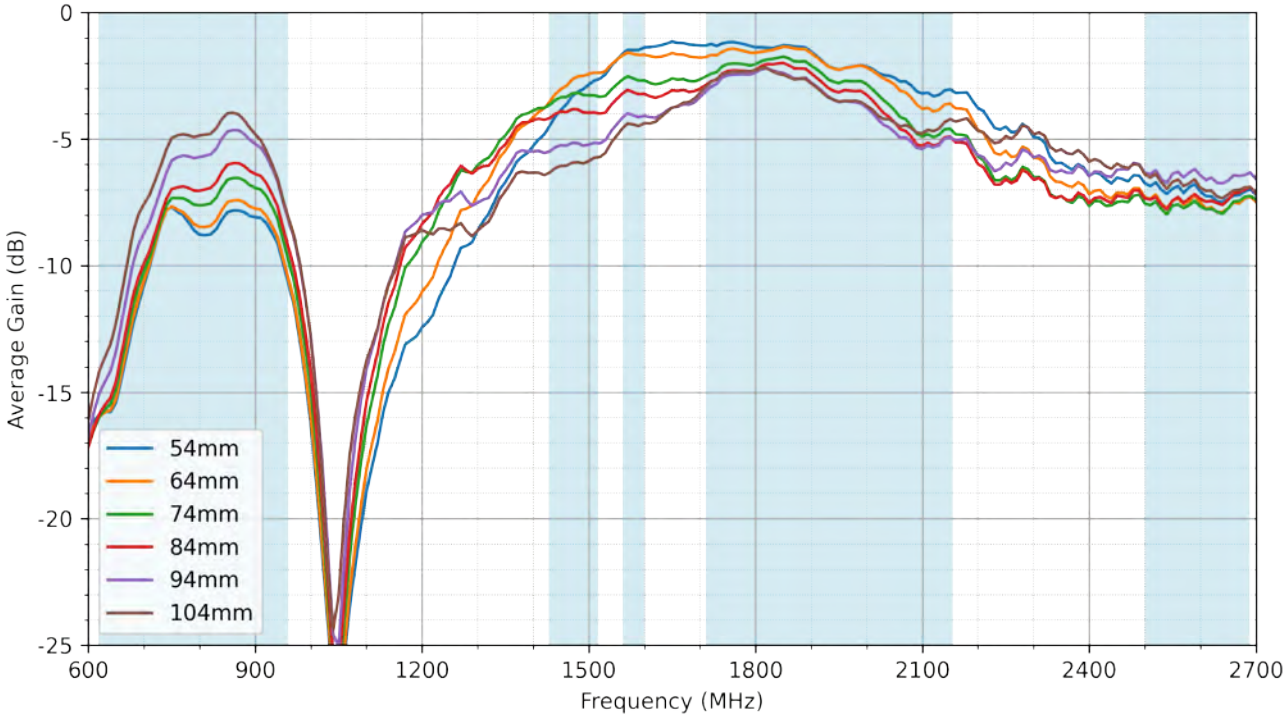
8.3 Peak Gain – North America



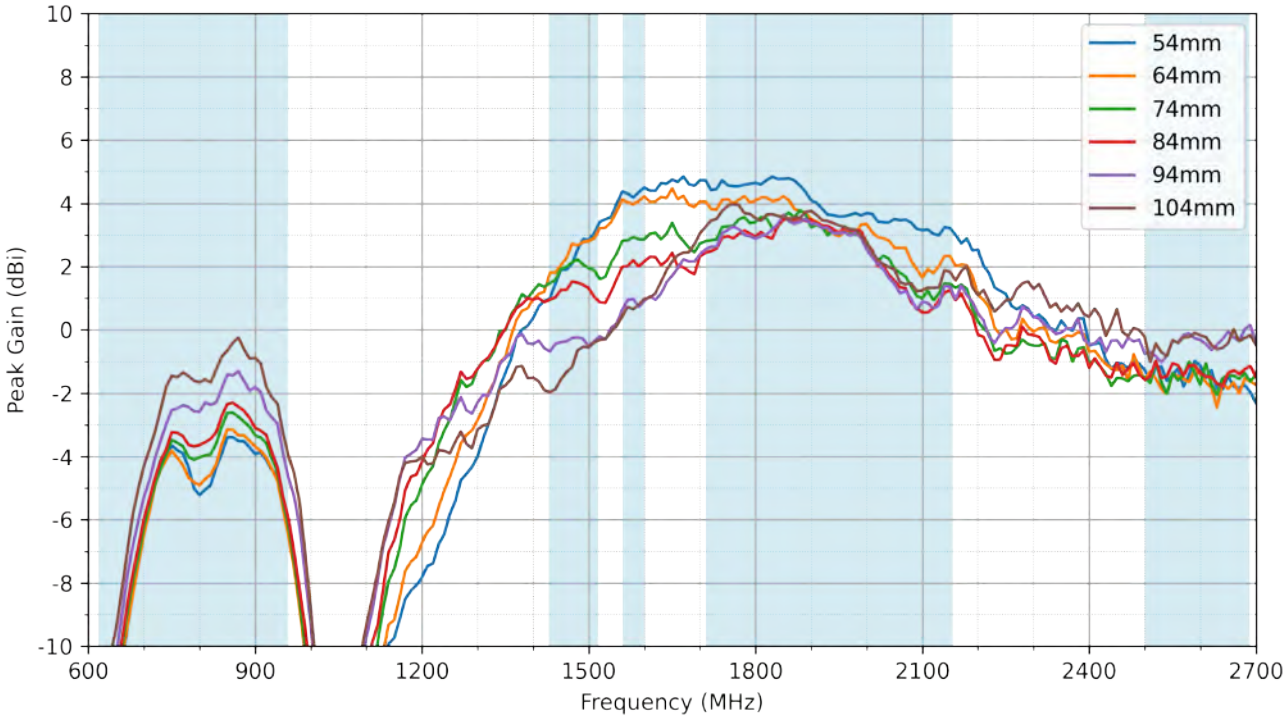
8.4 Efficiency – EMEA



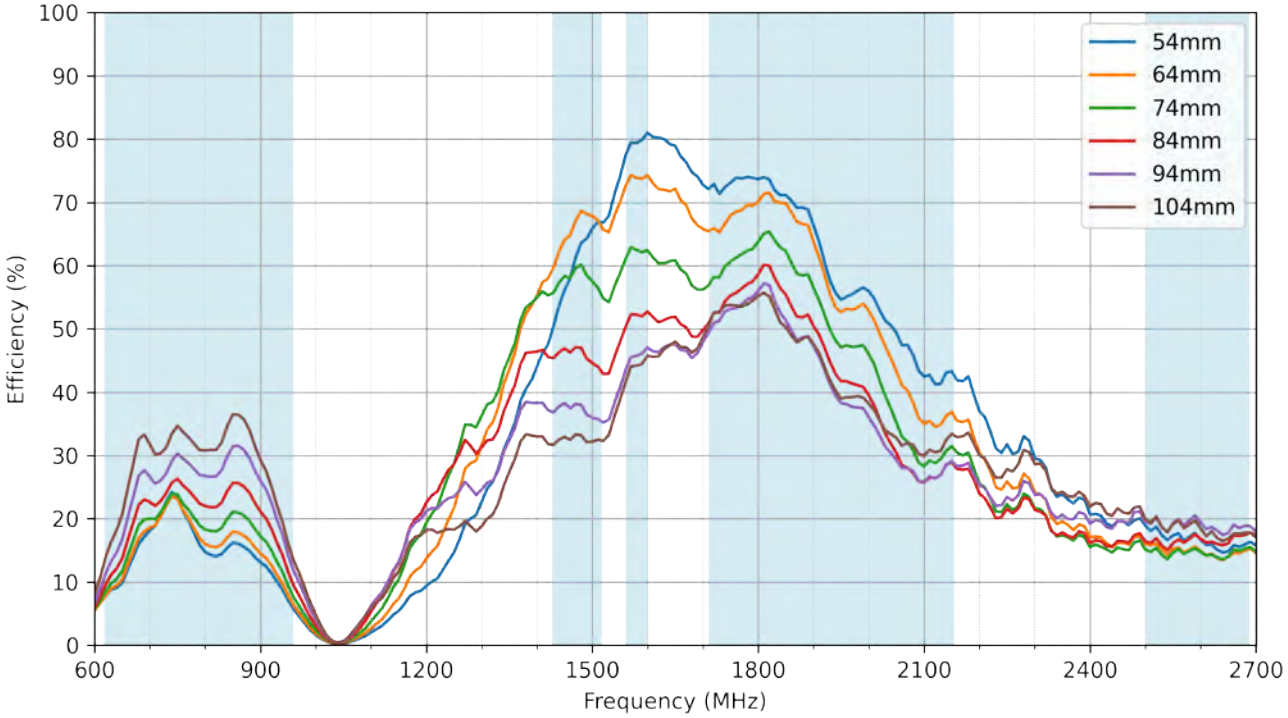
8.5 Average Gain – EMEA



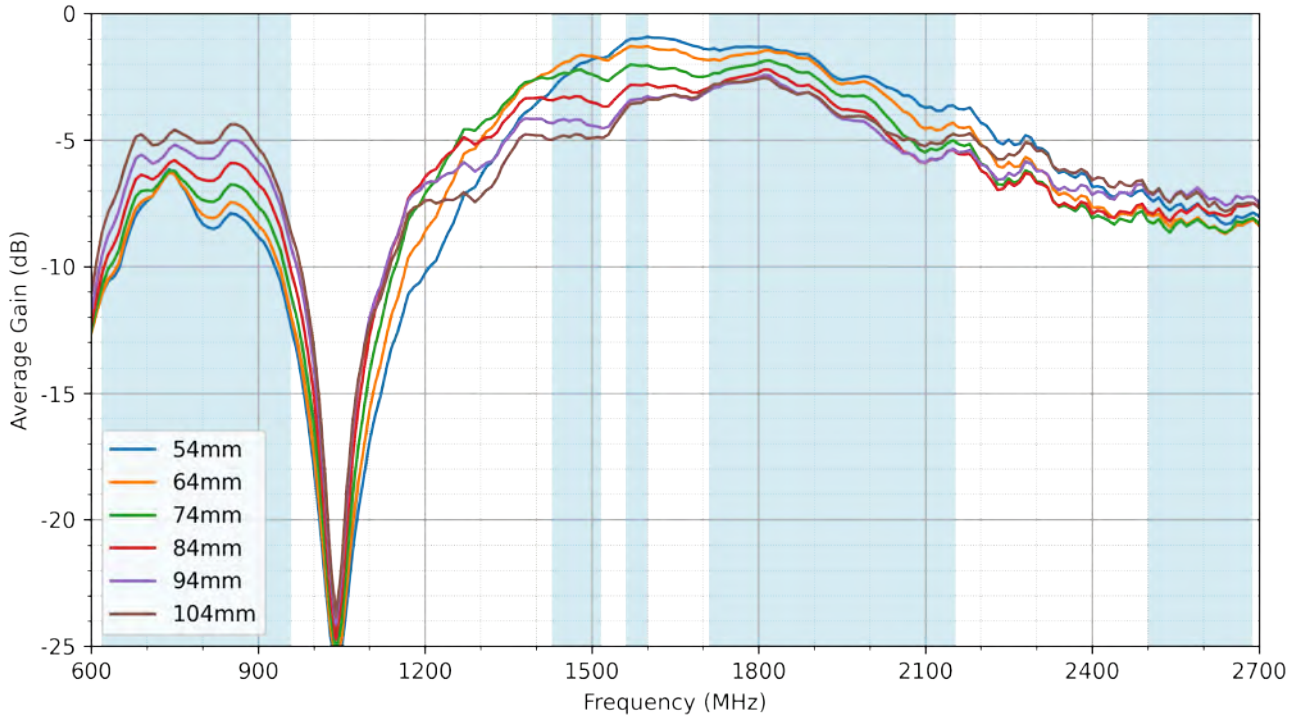
8.6 Peak Gain – EMEA



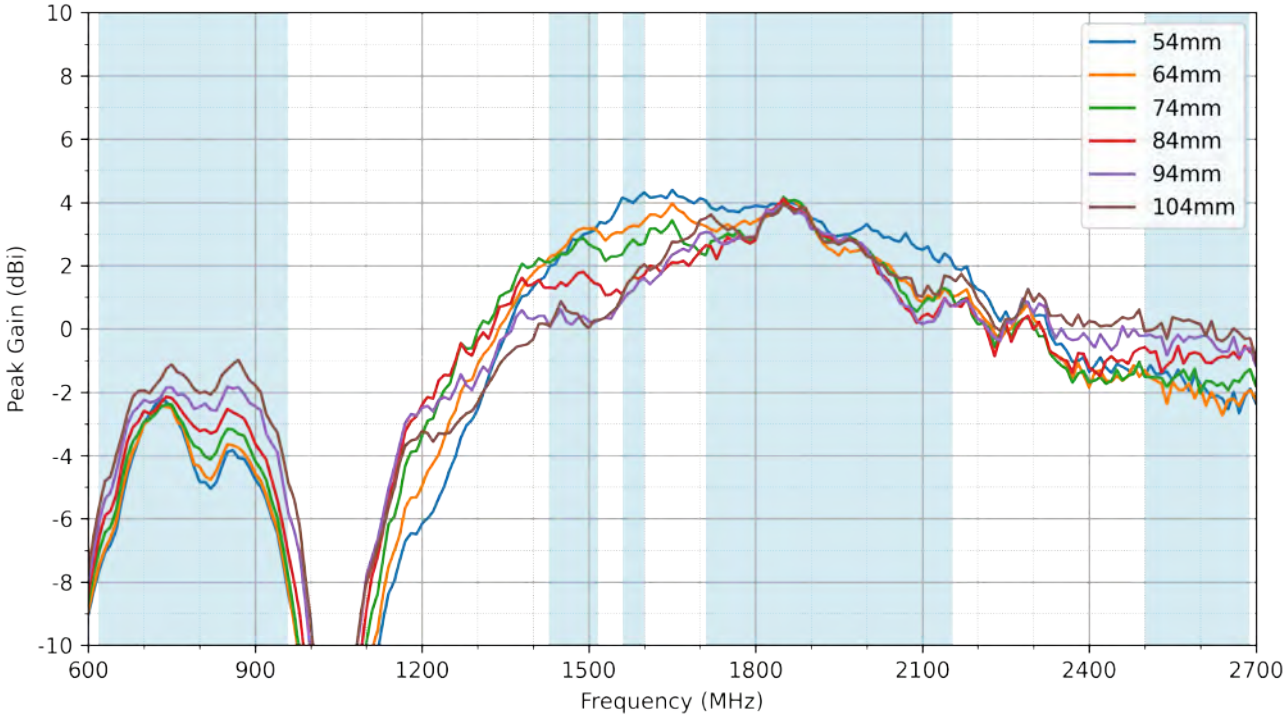
8.7 Efficiency – Worldwide



8.8 Average Gain – Worldwide

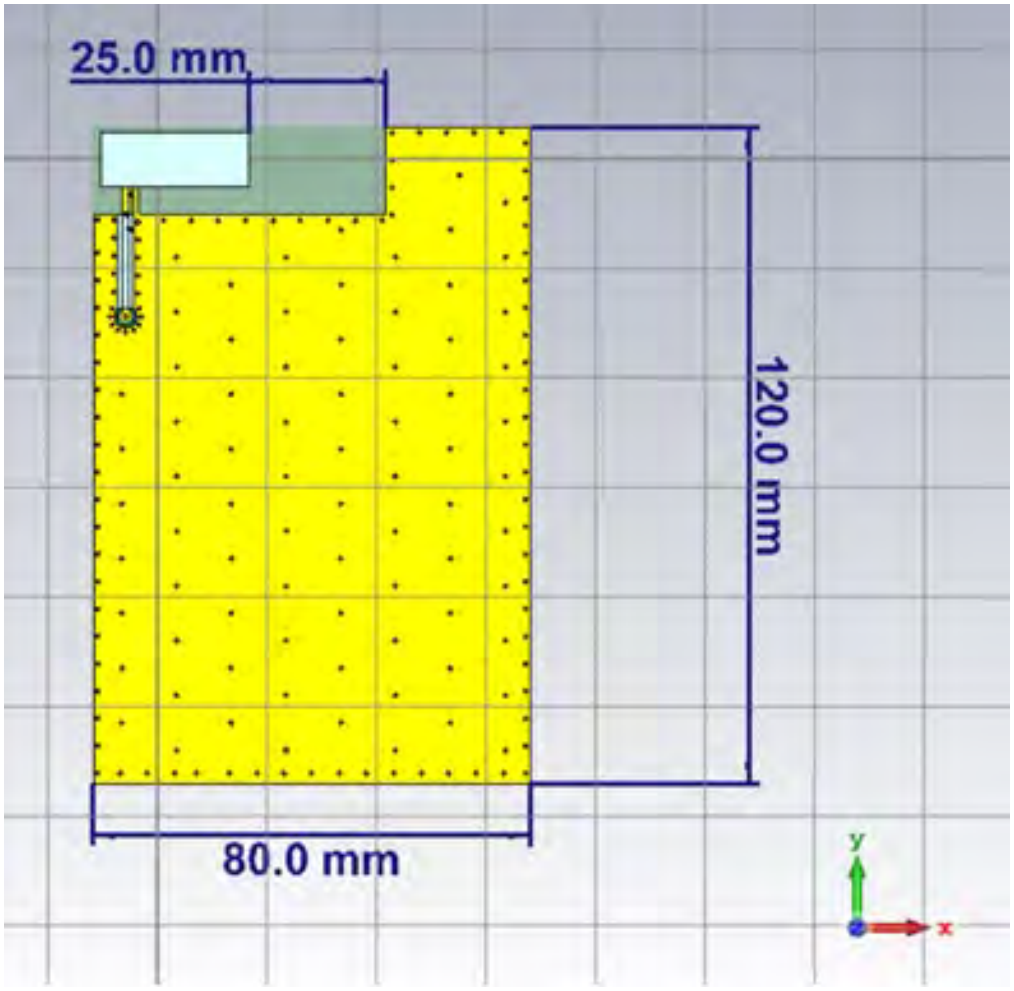


8.9 Peak Gain – Worldwide



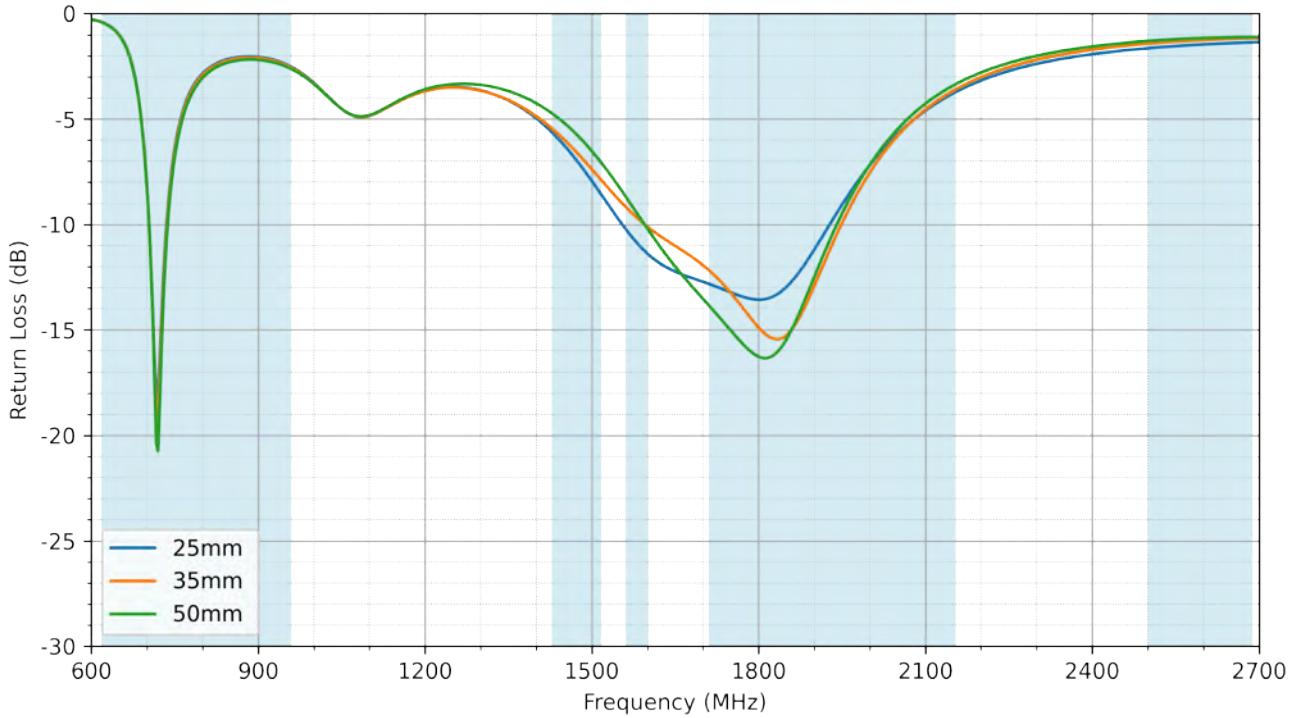
8.10 Effects of Right Side PCB Ground on Antenna Performance

The PCS.55.A antenna was tuned for a 120x80mm ground plane and the distance between the PCB ground on the right side of the antenna was parameterized and swept from 0 to 50 mm. The minimum condition, or 0mm, has the ground right up against the antenna and the maximum condition, or 50mm, has no ground on the right side of the antenna (i.e. a full 80x16mm keep out area). This was done in order to determine the minimum distance that the ground can be placed next to the antenna without affecting performance. The configuration with 25mm is shown below.

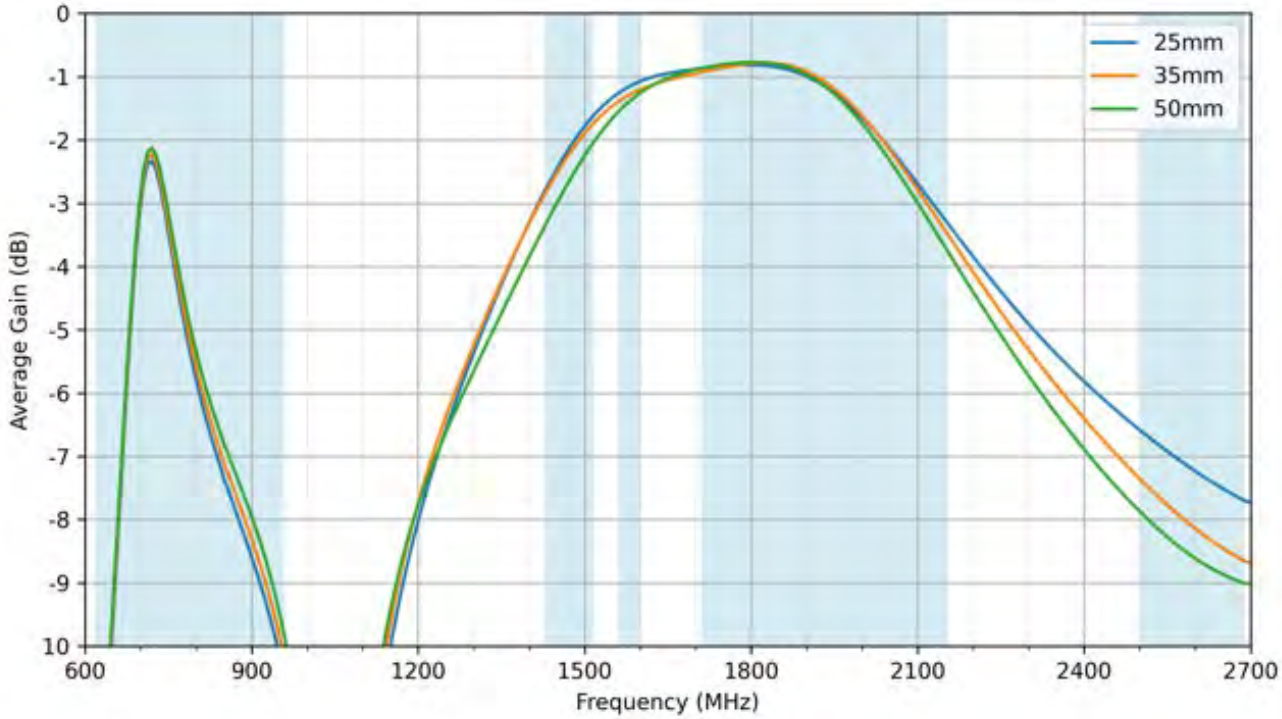


For minimal (0.2 dB) performance impact, a clearance of 25mm is recommended. Return loss and efficiency 25, 35 and 50mm clearance are shown below:

8.10.1 Return Loss

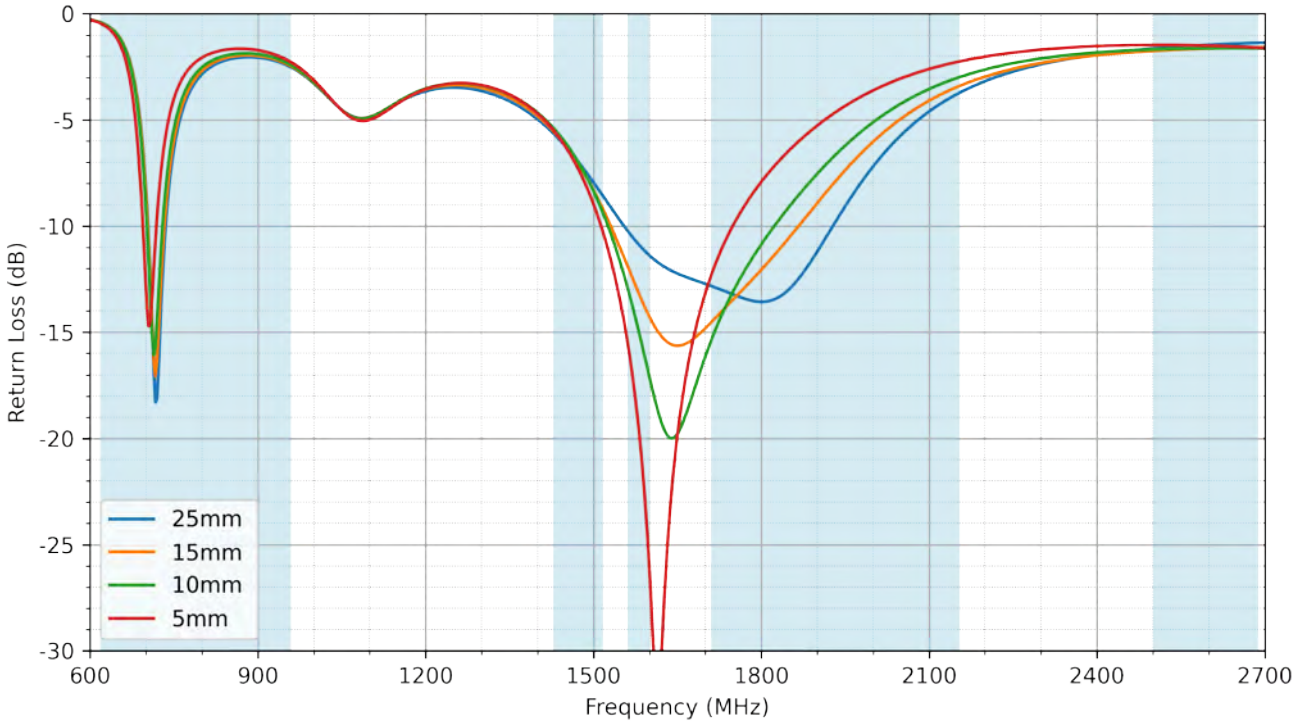


8.10.2 Efficiency

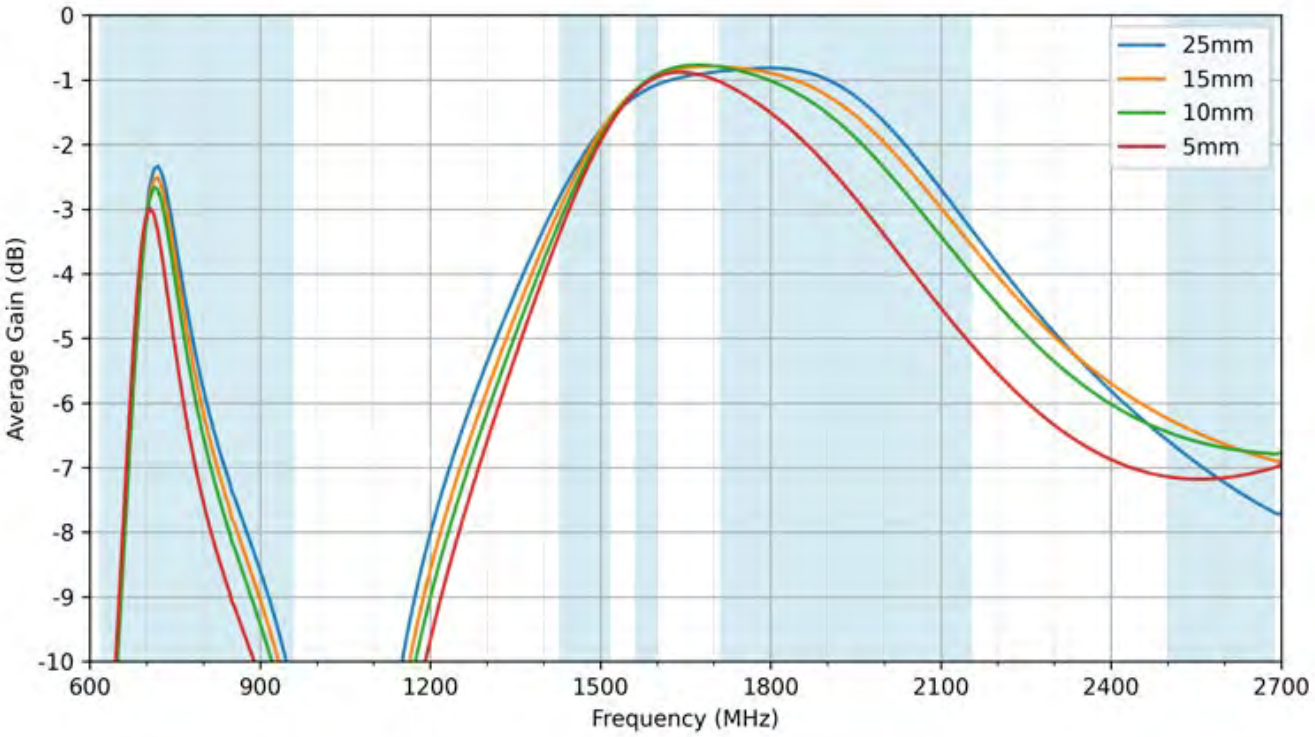


8.10.3 Return Loss

For more compact designs, a distance of 5 or 10mm can be implemented at the cost of efficiency and bandwidth as shown below:



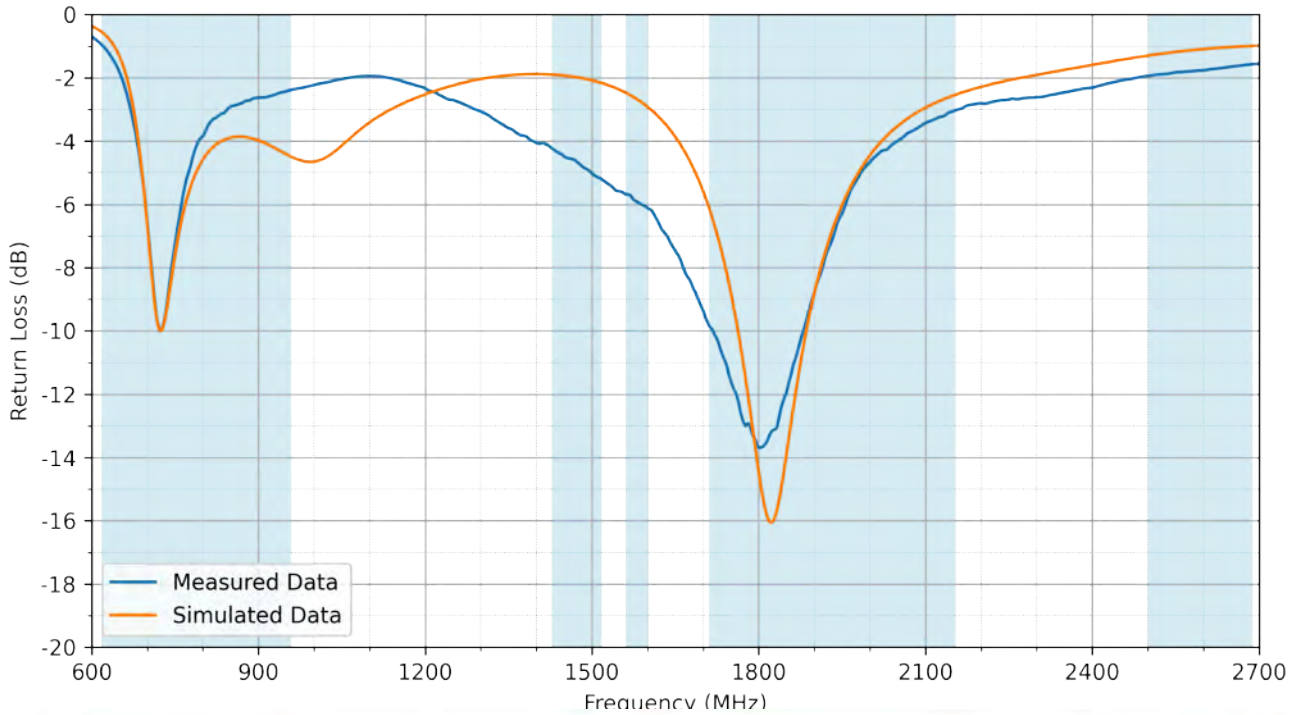
8.10.4 Efficiency

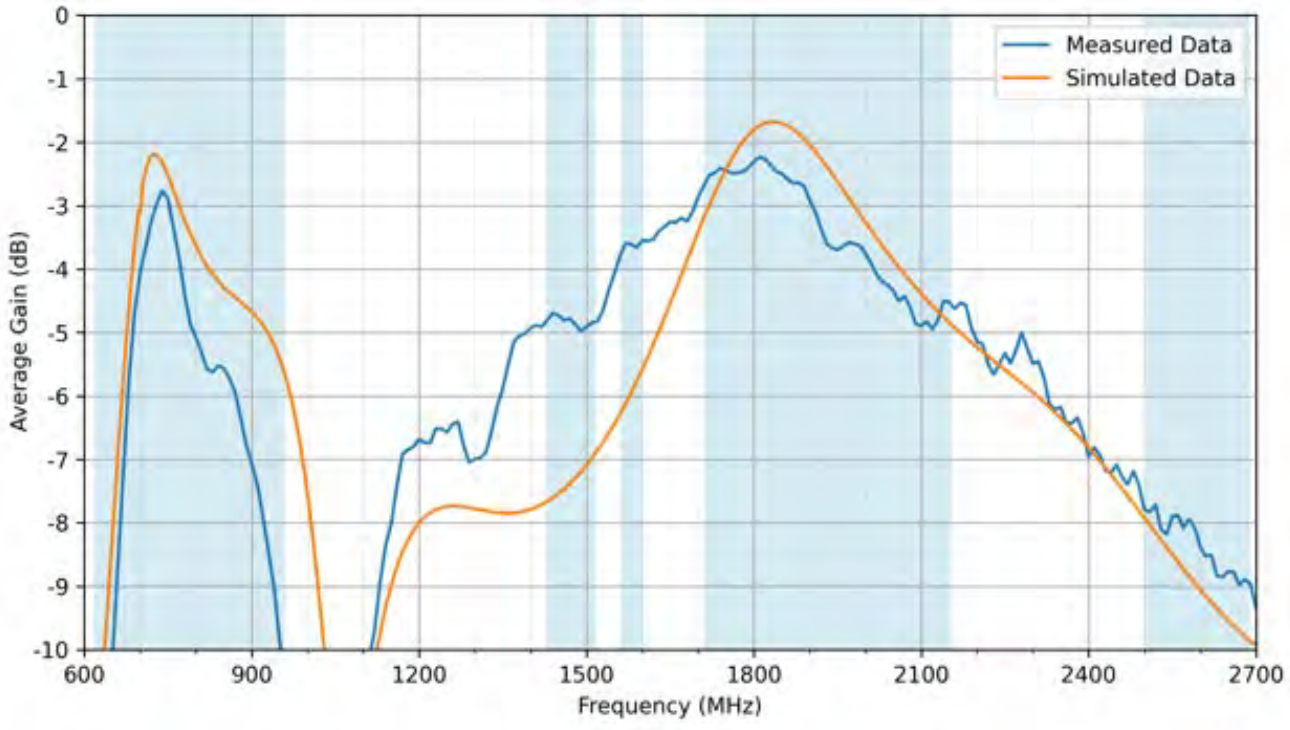


8.11 Correlation of CST Blackbox Model to Measured Results

To evaluate the accuracy of the CST Blackbox Model available to all customers, the following comparisons of return loss and efficiency have been made:

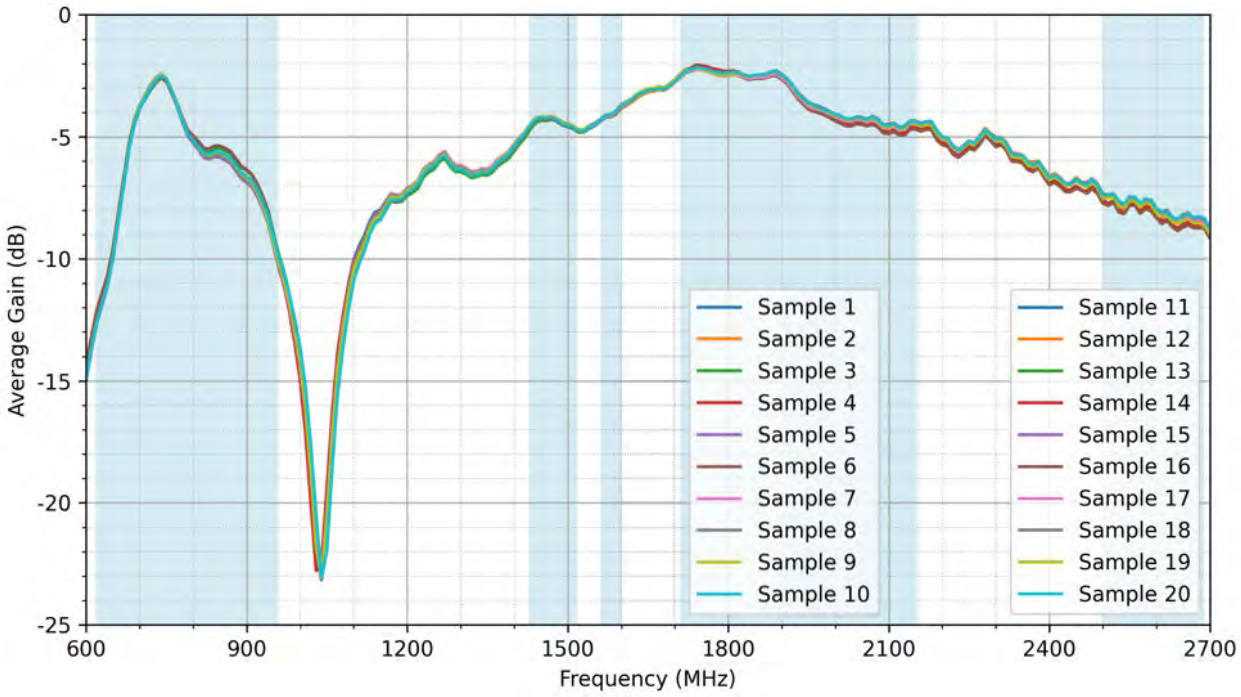
Please note the following simulated results were run at 18 cells/wavelength and with a -80 dB accuracy criterion. The measured results are for the PCS.55.A evaluation board with the NA tuning in place.





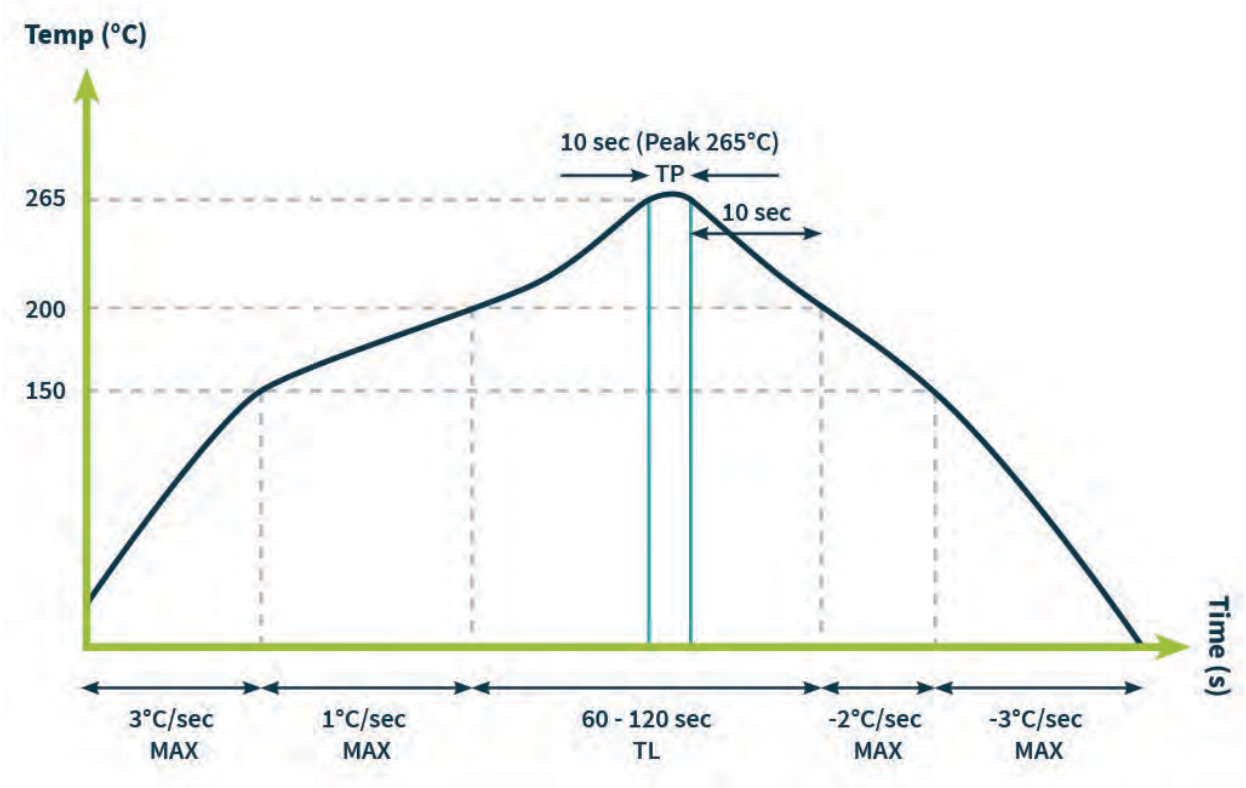
8.12 Repeatability Study

In order to verify the repeatability of the design, 20 production samples of the evaluation board for the PCS.55.A were tested in an anechoic chamber on the same day with standard SMA torque of 1 N·m



9 Solder Reflow Profile

The PCS.55.A can be assembled by following the recommended soldering temperatures are as follows:



*Temperatures listed within a tolerance of +/- 10º C

The PCS.55.A is not limited to the number of passes through the reflow process. Smaller components are typically mounted on the first pass, however, we do advise mounting the PCS.55.A when placing larger components on the board during subsequent reflows.

Changelog for the datasheet

SPE-23-8-012 – PCS.55.A

Revision: A (First Release)	
Date:	2023-02-10
Changes:	First Release
Changes Made by:	Tim Kelley

Previous Revisions

TAOGLAS®

www.taoglas.com