



TAOGLAS®



Datasheet

Part No:
PA.11

Description

2.4GHz Band Dielectric Ceramic PIFA
SMT Antenna for Bluetooth/WLAN/ZigBee Applications

Features:

2400-2484Mhz, 1.5dBi Peak Gain
Size: 10mm*4mm*3mm
Designed for the top right hand corner edge of the Component side
of the board (bottom left corner edge)
SMT Mount
RoHS & Reach Compliant

1.	Introduction	2
2.	Specification	3
3.	Antenna Characteristics	4
4.	Radiation Patterns	7
5.	Mechanical Drawing	11
6.	Antenna Integration Guide	12
7.	Packaging	19
8.	Solder Reflow Profile	20
<hr/>		
	Changelog	21

Taoglas makes no warranties based on the accuracy or completeness of the contents of this document and reserves the right to make changes to specifications and product descriptions at any time without notice. Taoglas reserves all rights to this document and the information contained herein. Reproduction, use or disclosure to third parties without express permission is strictly prohibited.



1. Introduction



This specification covers the Dielectric PIFA Antenna for 2400-2484MHz, covering such applications as Wi-Fi, Bluetooth and Zigbee. A ceramic dielectric

PIFA antenna offers smallest footprint, superior gain characteristics and improved isolation over traditional PCB based antennas. This antenna has been developed for the top right hand corner edge of the component side of the Board (bottom left corner edge), the antenna has to be positioned on a non-ground (copper/metal free) area with the feed-point matched direct to the module. Please refer to Recommended Foot print Diagram (8.0 Page 14).

Many module manufacturers specify peak gain limits for any antennas that are to be connected to that module. Those peak gain limits are based on free-space conditions. In practice, the peak gain of an antenna tested in free-space can degrade by at least 1 or 2dBi when put inside a device. So ideally you should go for a slightly higher peak gain antenna than mentioned on the module specification to compensate for this effect, giving you better performance.

Upon testing of any of our antennas with your device and a selection of appropriate layout, integration technique, or cable, Taoglas can make sure any of our antennas' peak gain will be below the peak gain limits. Taoglas can then issue a specification and/or report for the selected antenna in your device that will clearly show it complying with the peak gain limits, so you can be assured you are meeting regulatory requirements for that module.

For example, a module manufacturer may state that the antenna must have less than 2dBi peak gain, but you don't need to select an embedded antenna that has a peak gain of less than 2dBi in free-space. This will give you a less optimized solution. It is better to go for a slightly higher free-space peak gain of 3dBi or more if available. Once that antenna gets integrated into your device, performance will degrade below this 2dBi peak gain due to the effects of GND plane, surrounding components, and device housing. If you want to be absolutely sure, contact Taoglas and we will test. Choosing a Taoglas antenna with a higher peak gain than what is specified by the module manufacturer and enlisting our help will ensure you are getting the best performance possible without exceeding the peak gain limits.

2. Specification

Wi-Fi Electrical

Band	Frequency (MHz)	Efficiency (%)	Average Gain (dB)	Peak Gain (dBi)	Impedance	Polarization	Radiation Pattern	VSWR
Wi-Fi - 2GHz	2400-2500	79.1	-1.02	5.02	50 Ω	Linear	Omni	2.0 Max

Mechanical

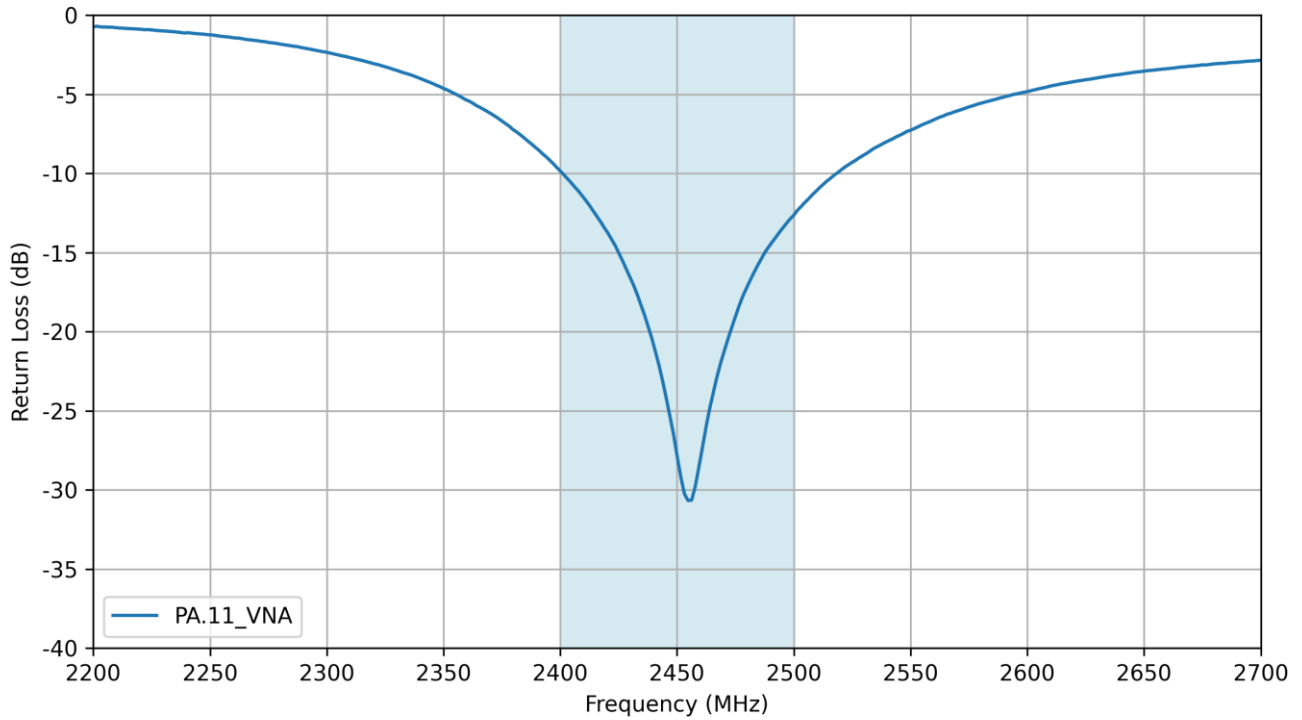
Dimensions	10*4*3mm
Termination	Ag (Environmentally Friendly Lead-Free)

Environmental

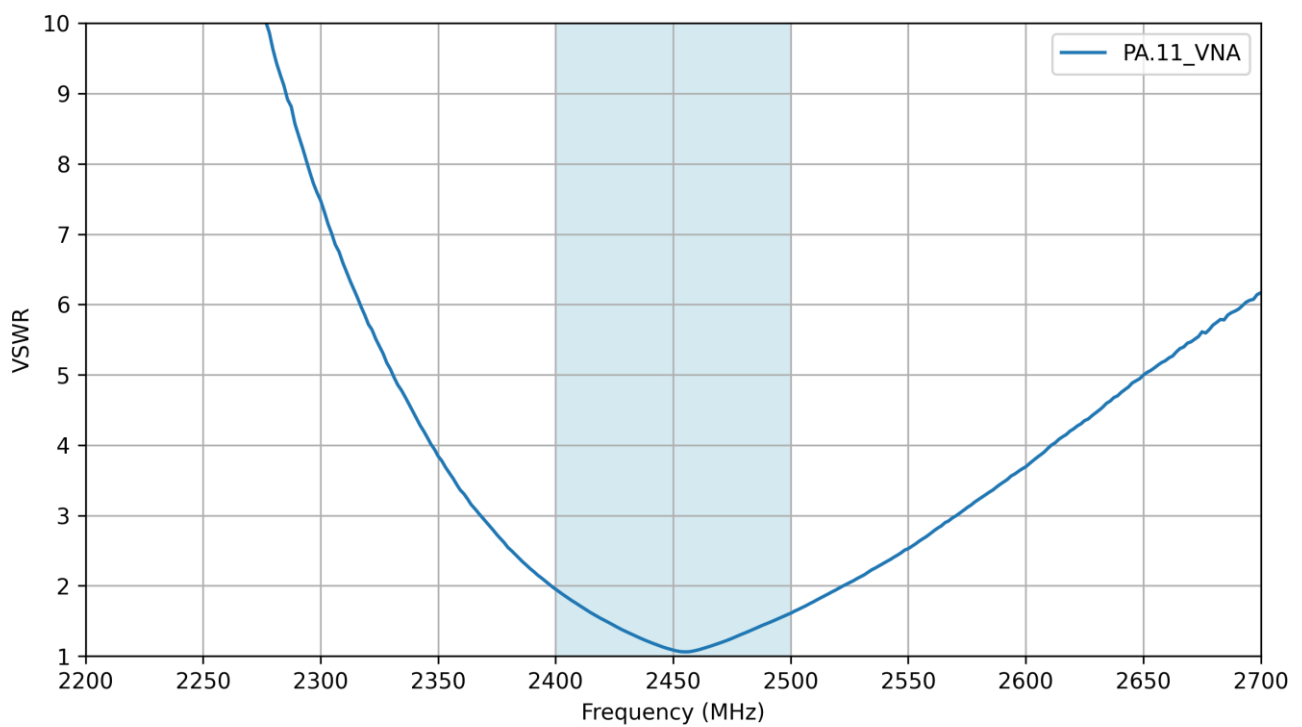
Operating Temperature	-40~+85°C
Moisture Sensitivity Level (MSL)	3 (168 Hours)

3. Antenna Characteristics

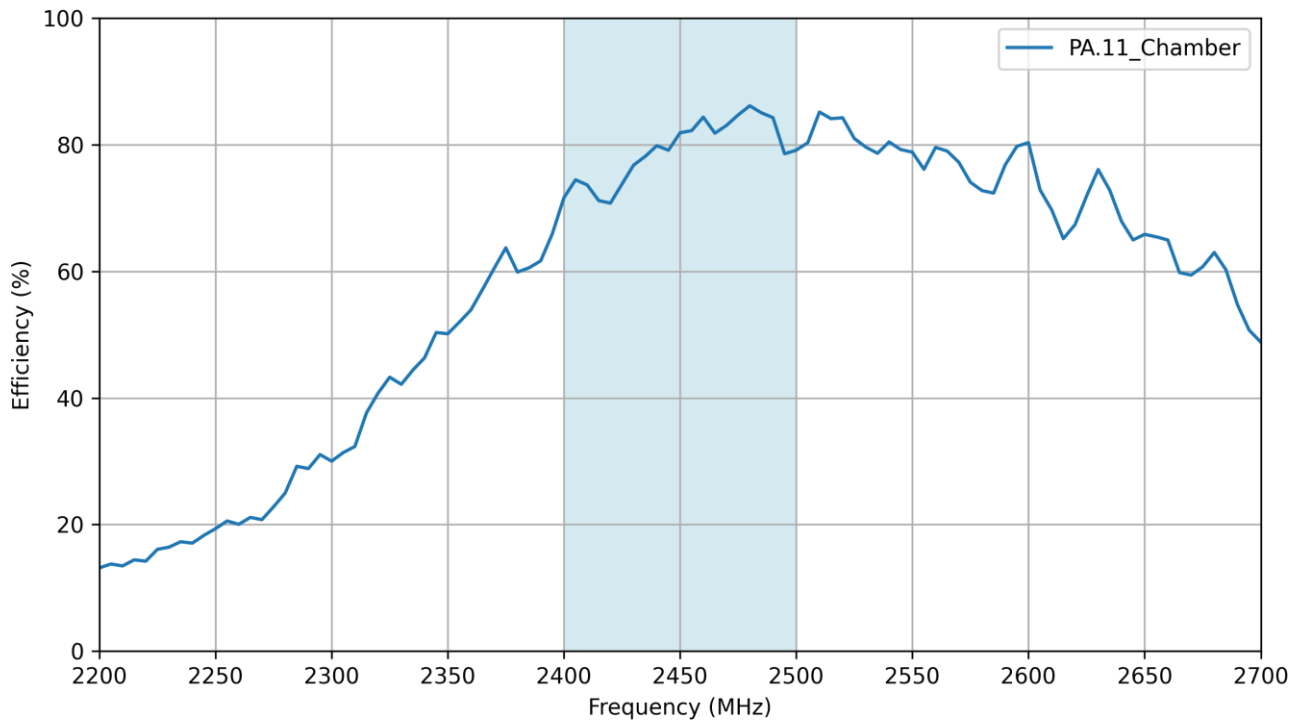
3.1 Return Loss



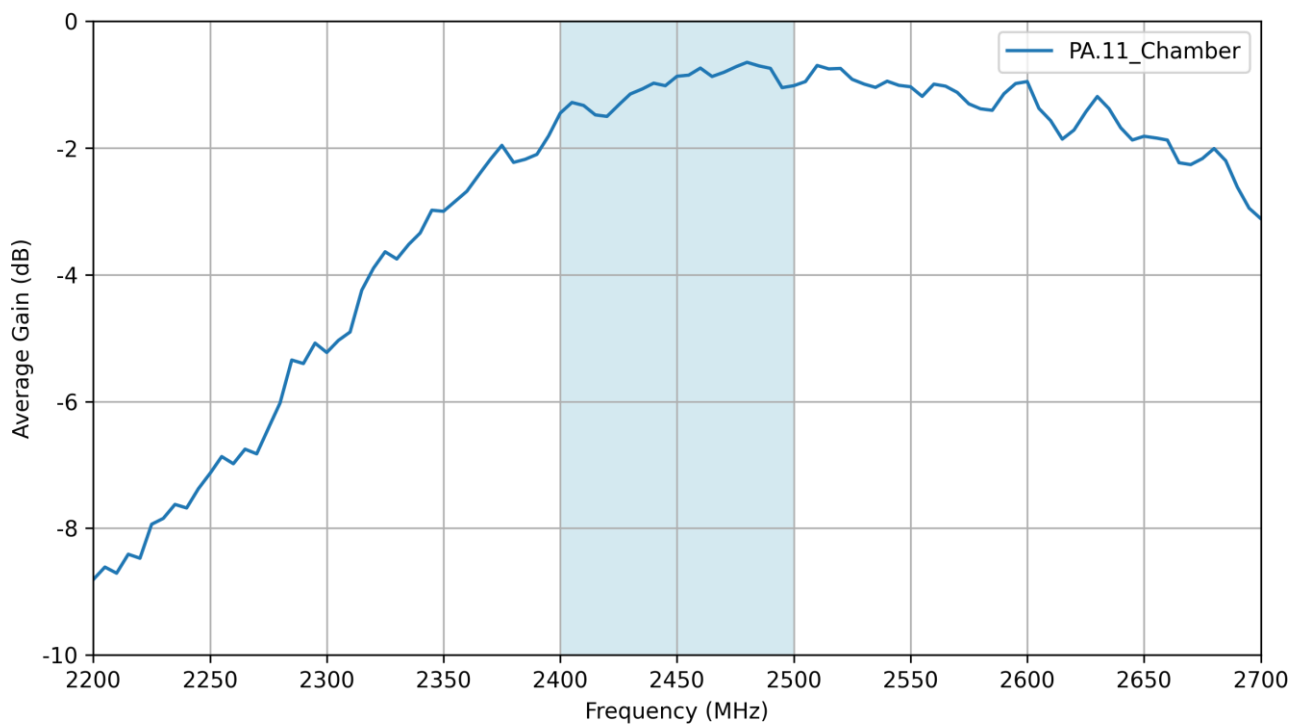
3.2 VSWR



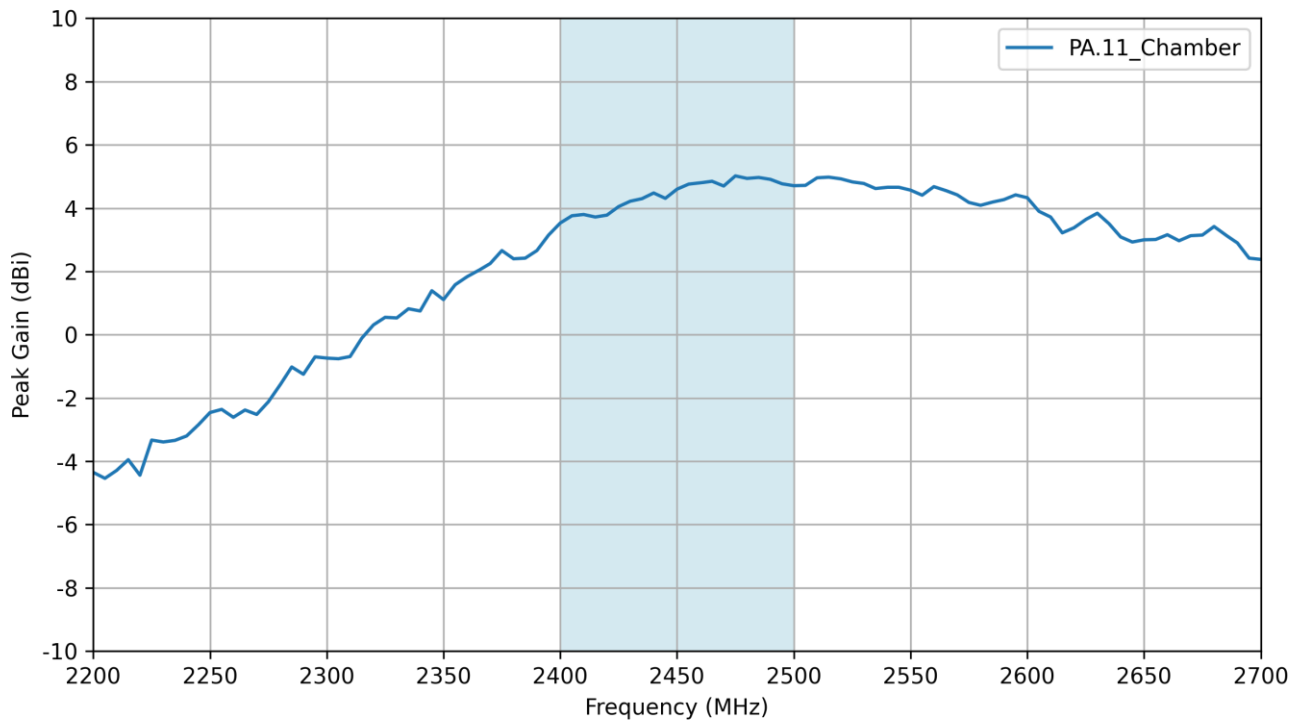
3.3 Efficiency



3.4 Average Gain

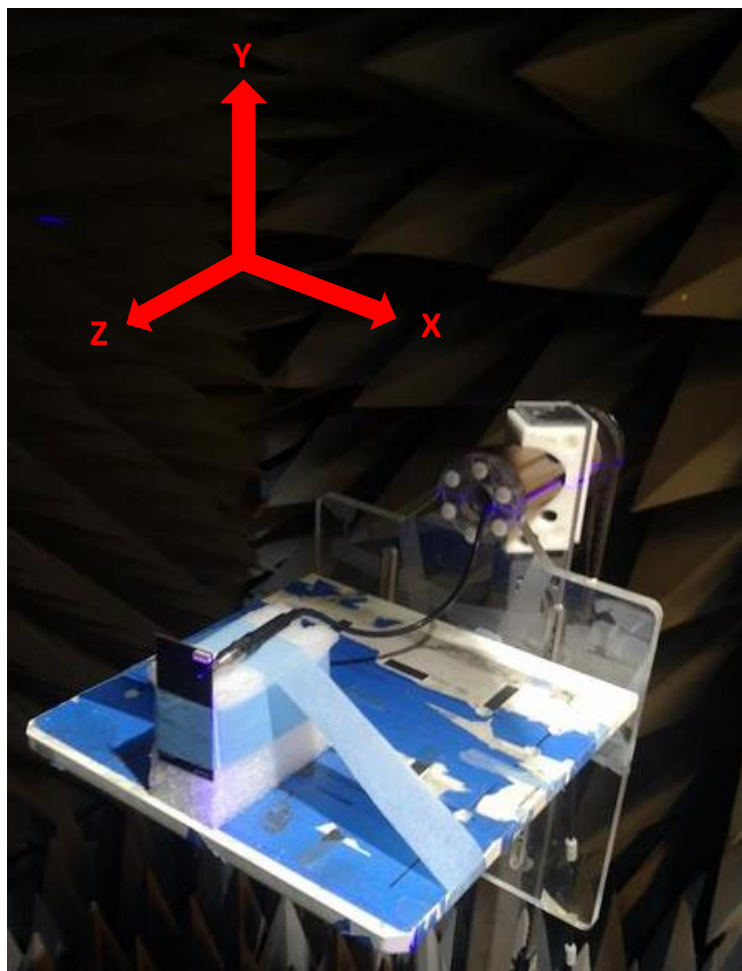
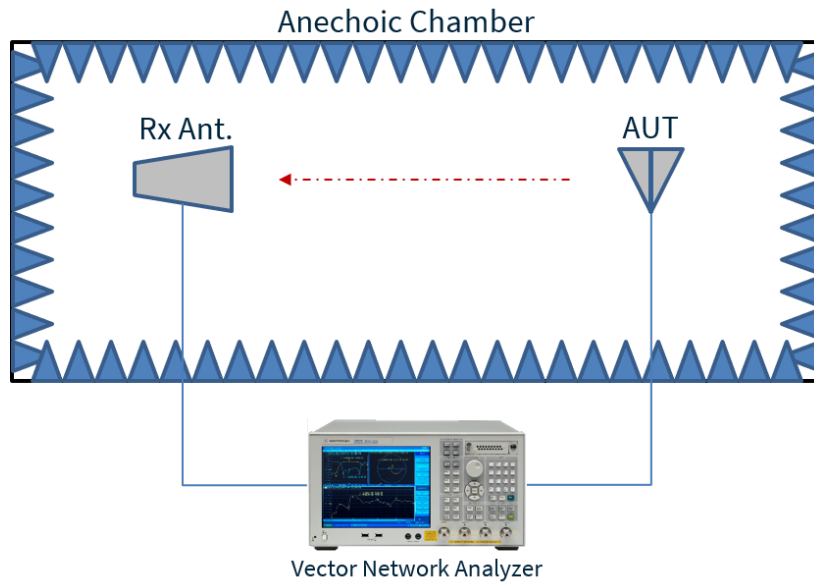


3.5 Peak Gain

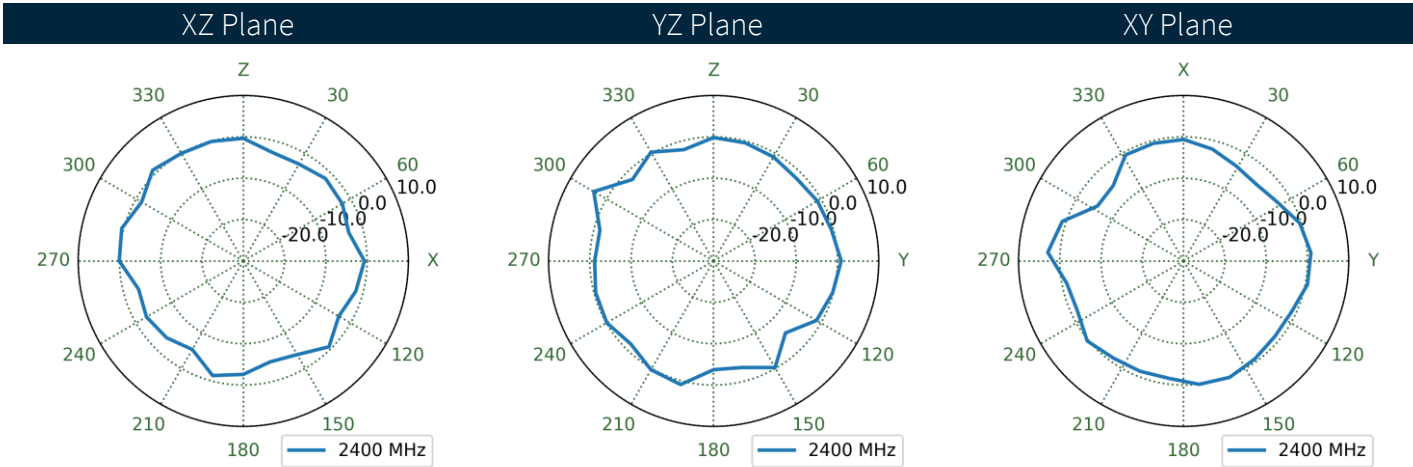
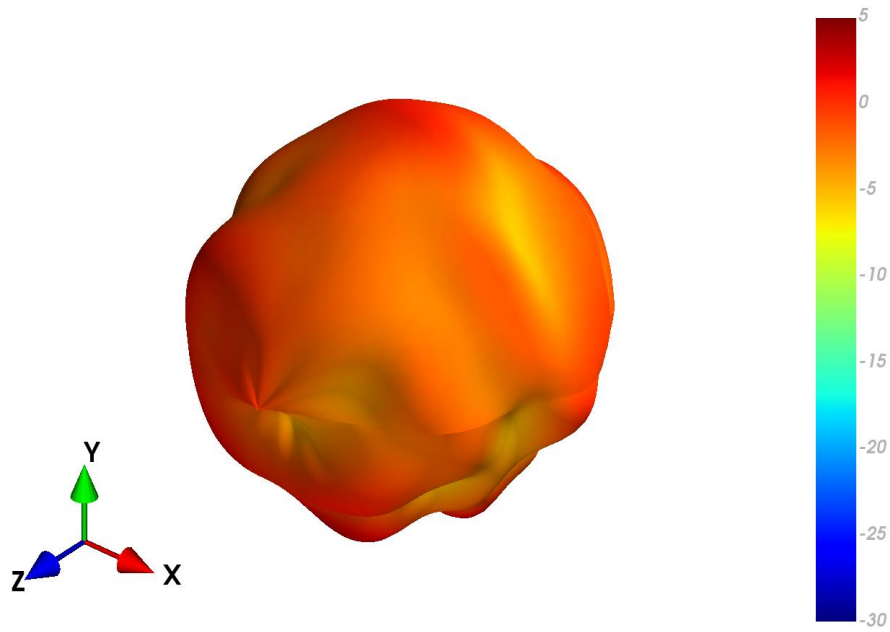


4. Radiation Patterns

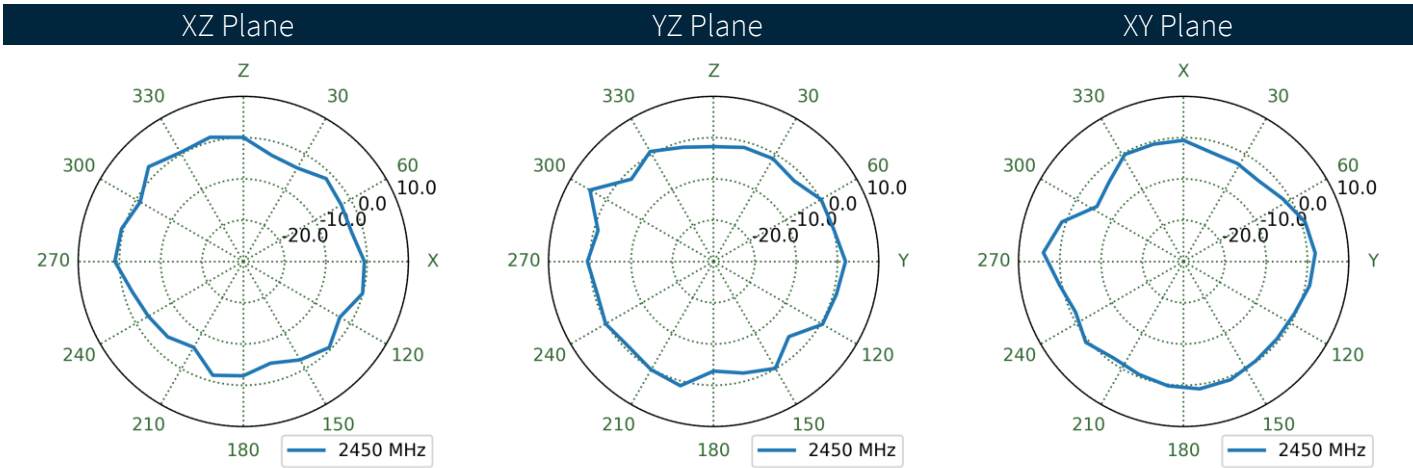
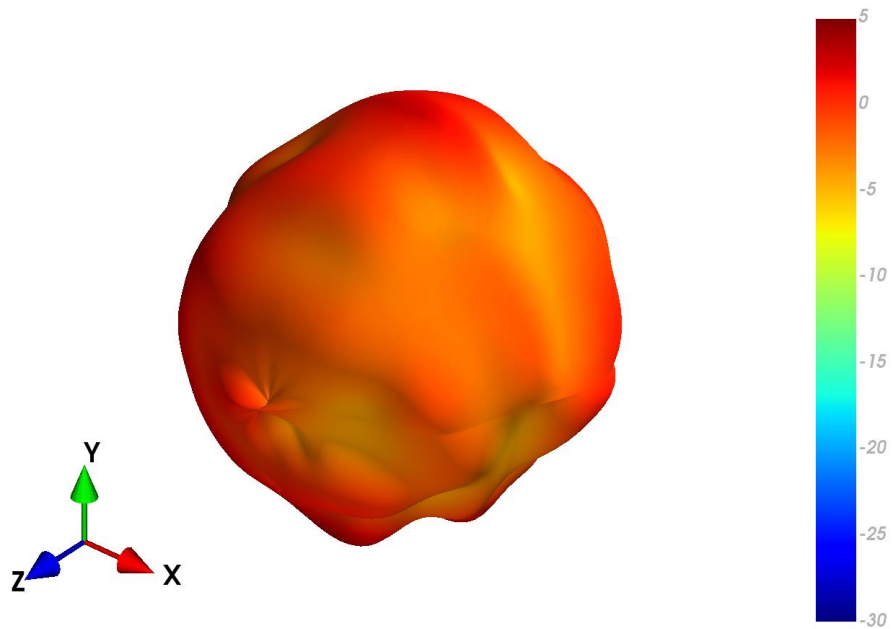
4.1 Test Setup



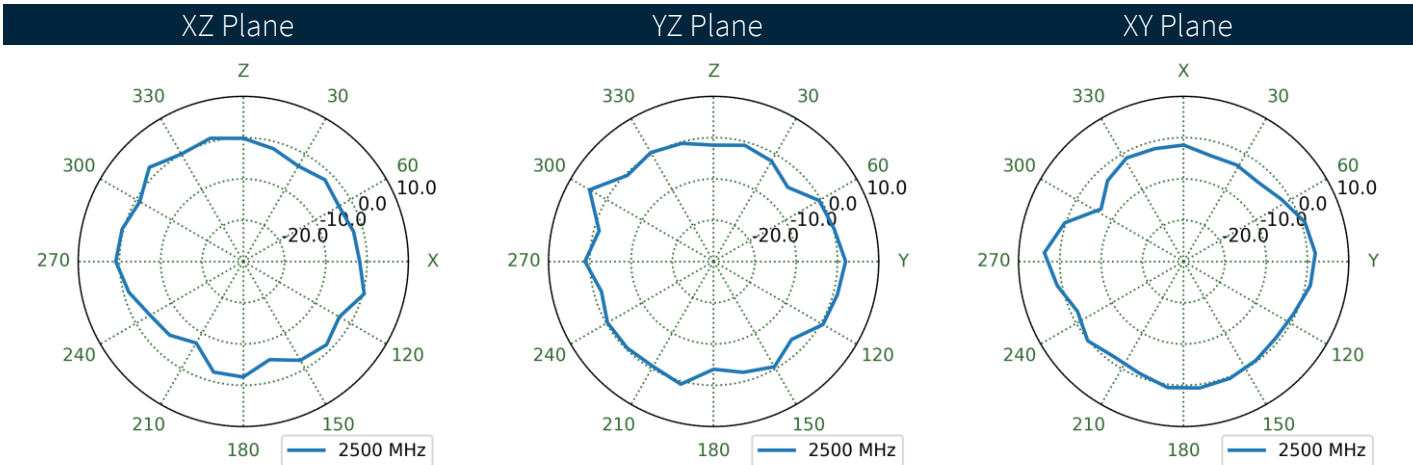
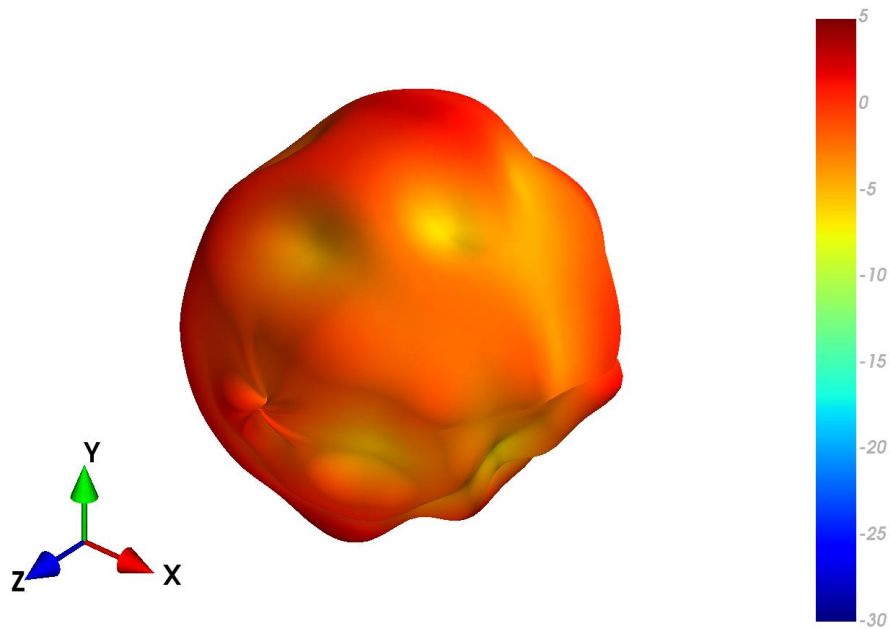
4.2 PA.11_Chamber Patterns at 2400 MHz



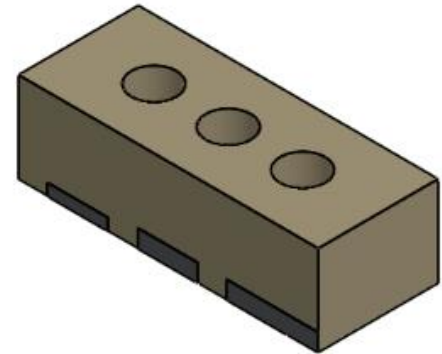
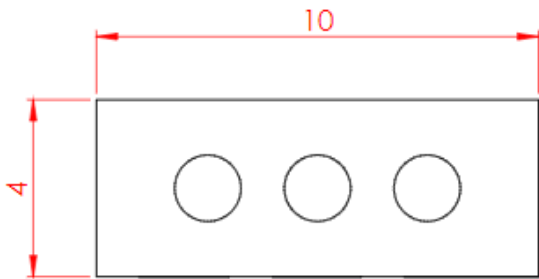
4.3 PA.11_Chamber Patterns at 2450 MHz



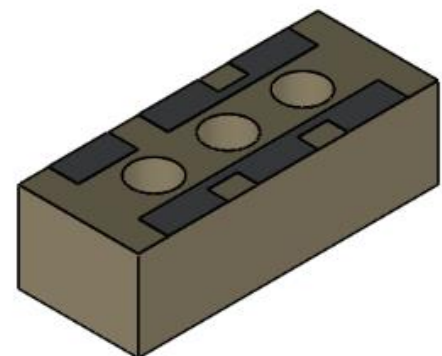
4.4 PA.11_Chamber Patterns at 2500 MHz



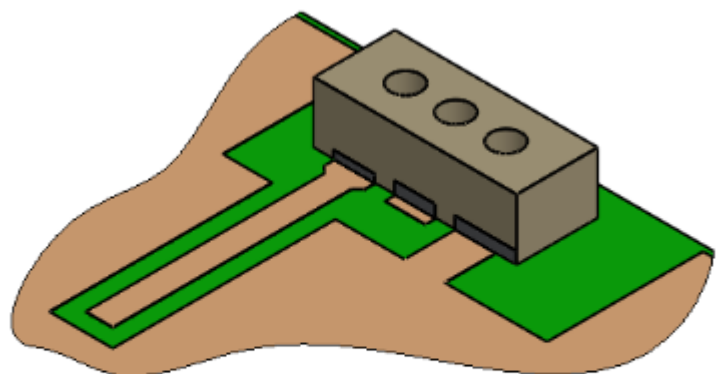
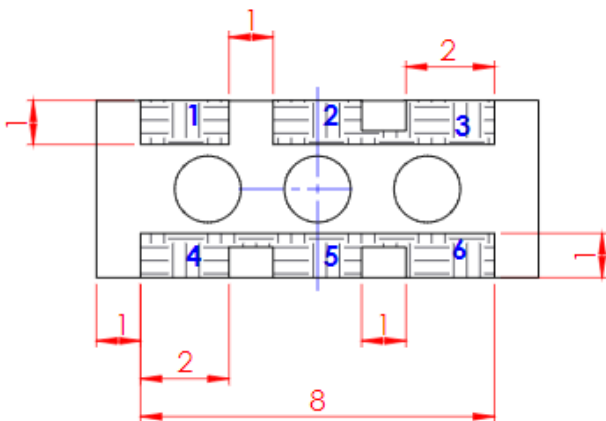
5. Mechanical Drawing



MODEL VIEW

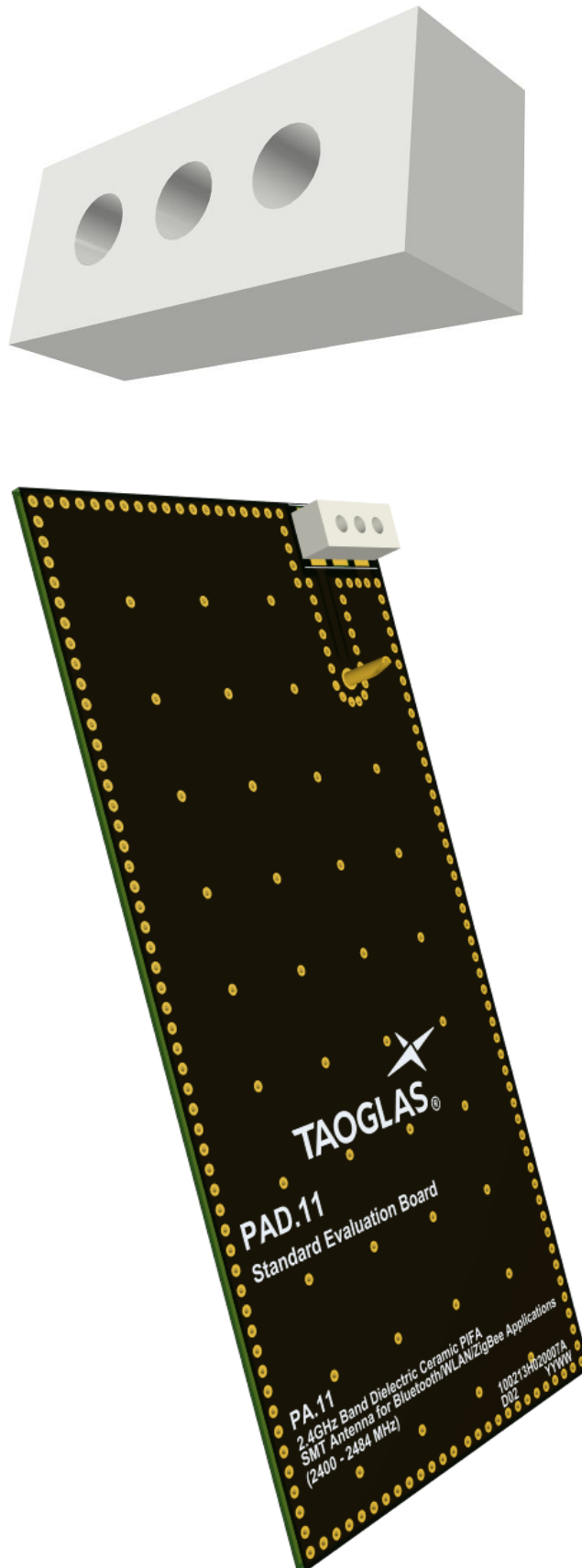


MODEL VIEW



ANTENNA ON FOOTPRINT PCB VIEW
SCALE 4:1

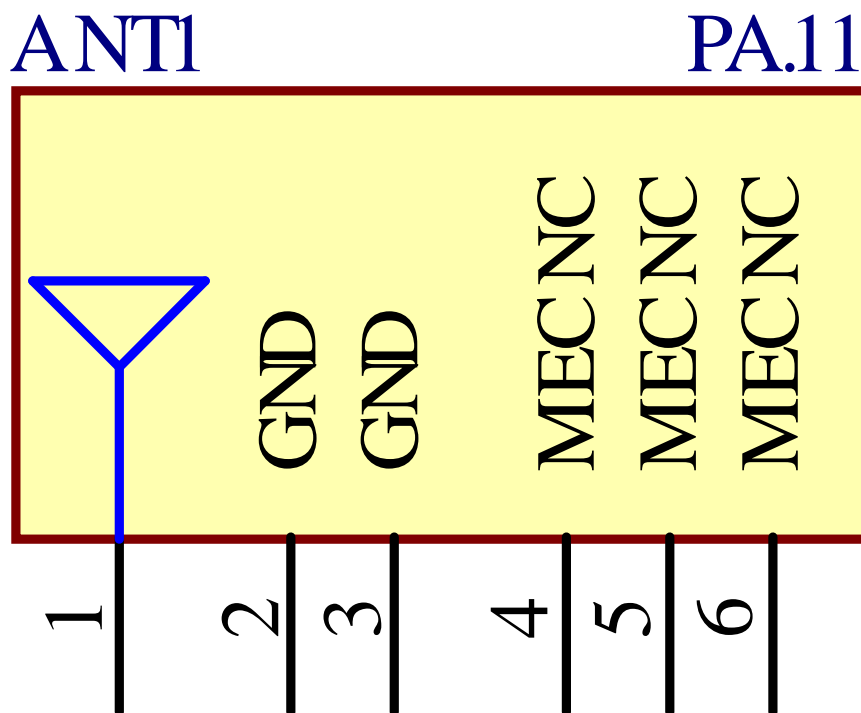
6. Antenna Integration Guide



6.1 Schematic and Symbol Definition

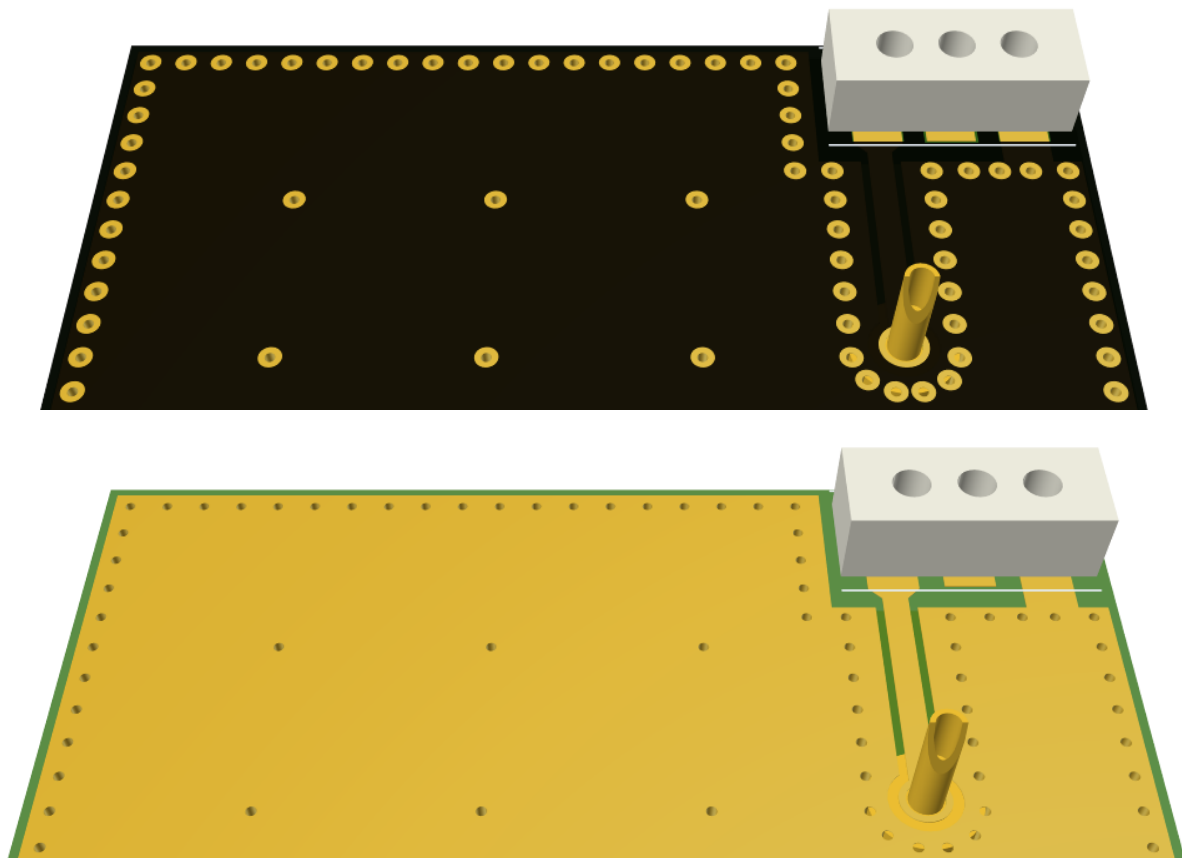
The circuit symbol for the antenna is shown below. The antenna has 6 pins with only three pins (Pin 1, Pin 2 and Pin 3) as functional. Pins 4, 5 and 6 are for mechanical strength.

Pin	Description
1	RF Feed
2, 3	Ground
4, 5, 6	Mechanical, Not Connected



6.2 Antenna Integration

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



6.3 PCB Layout

The footprint and clearance on the PCB must meet the layout drawing in (Footprint Drawing).



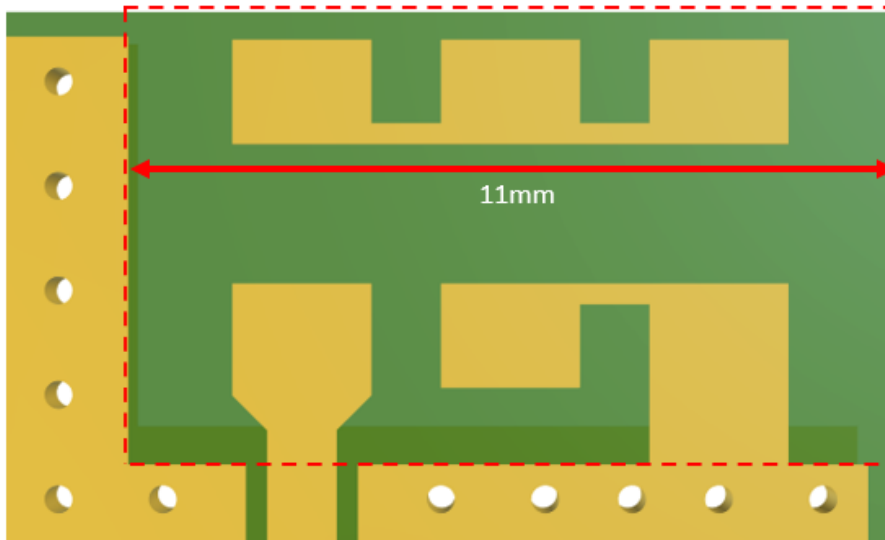
Topside



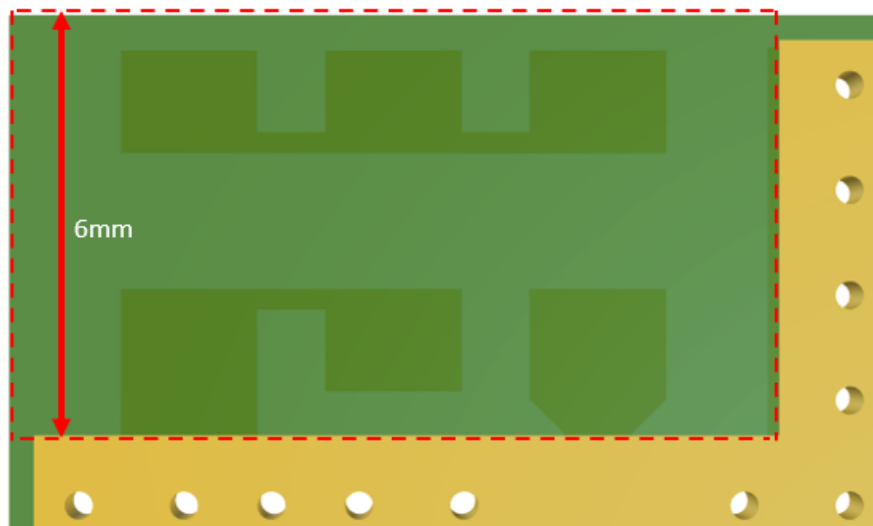
Bottom Side

6.4 PCB Keep-out

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 11mm in width and 6.5mm in length on the topside with 6mm in length on the bottom from the top right corner of the PCB. This clearance area includes the bottom side and ALL internal layers on the PCB.

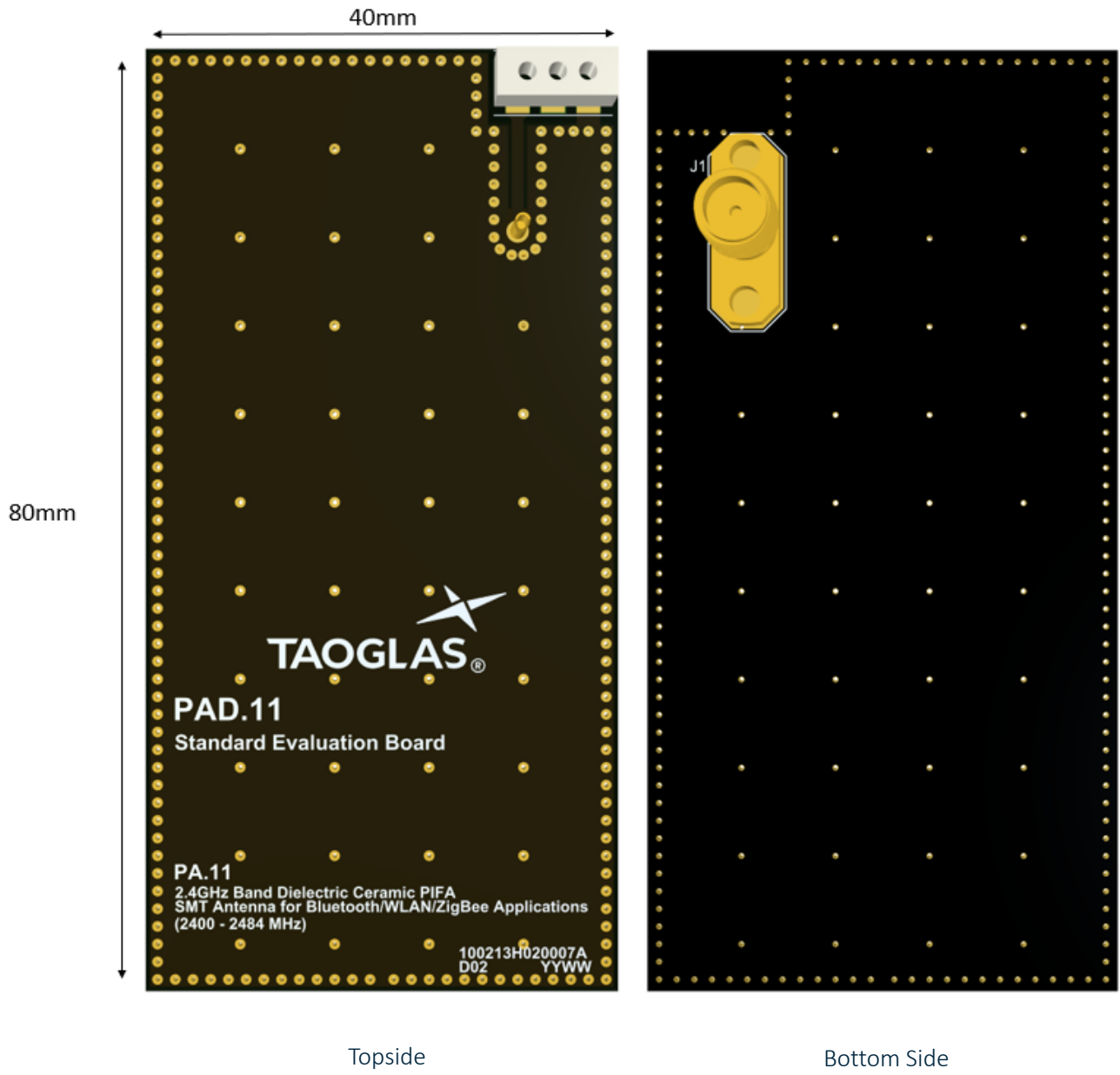


Topside

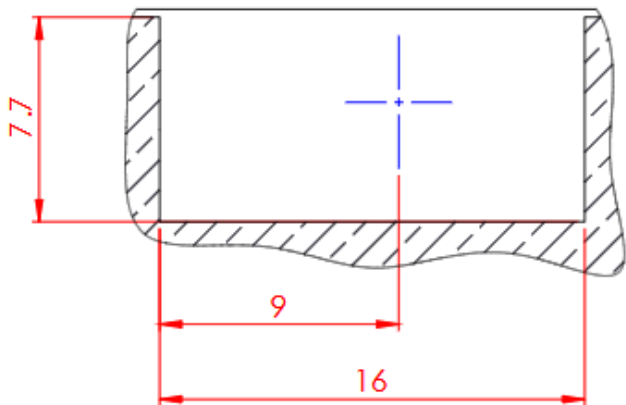
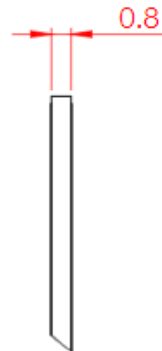
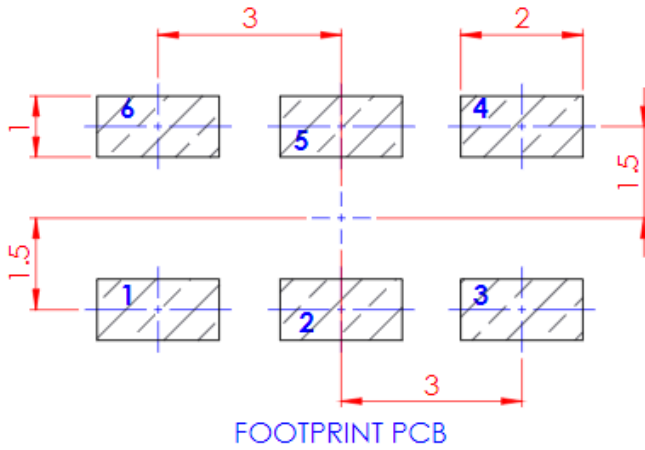


Bottom Side

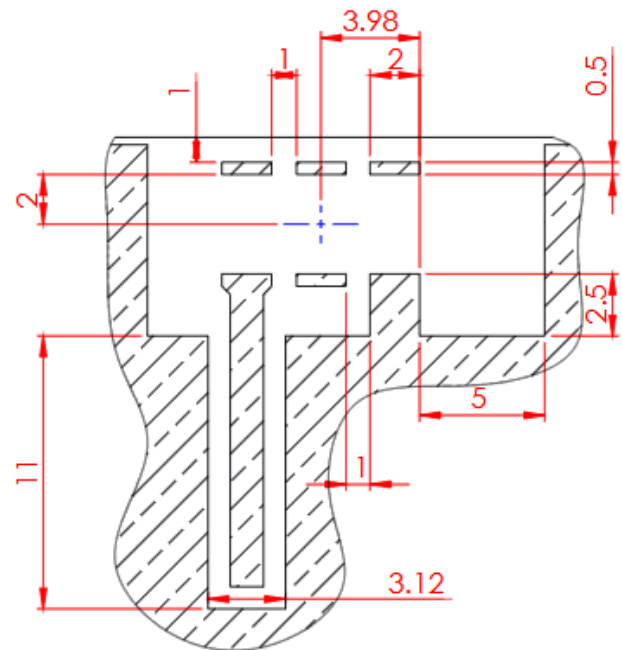
6.6 Evaluation Board



6.7 Footprint



GROUND CLEARANCE BOTTOM VIEW
(SILKSCREEN NOT SHOWN)

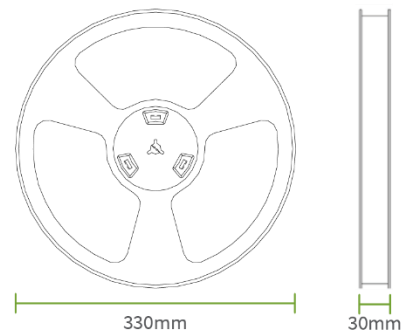


GROUND CLEARANCE TOP VIEW
(SILKSCREEN NOT SHOWN)

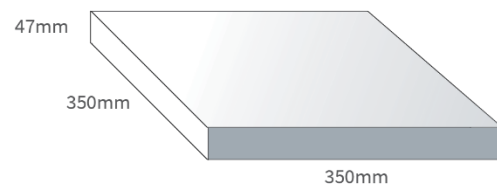
PIN	DESCRIPTION
1	RF FEED (50 Ohm)
2	MECHANICAL, NC
3	GROUND
4	MECHANICAL, NC
5	MECHANICAL, NC
6	MECHANICAL, NC

7. Packaging

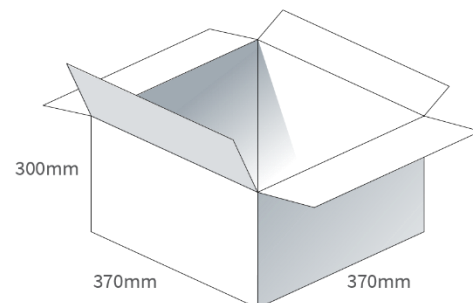
1000 pcs PA.11
 1 reel in small inner box
 Dimensions - 350*350*70



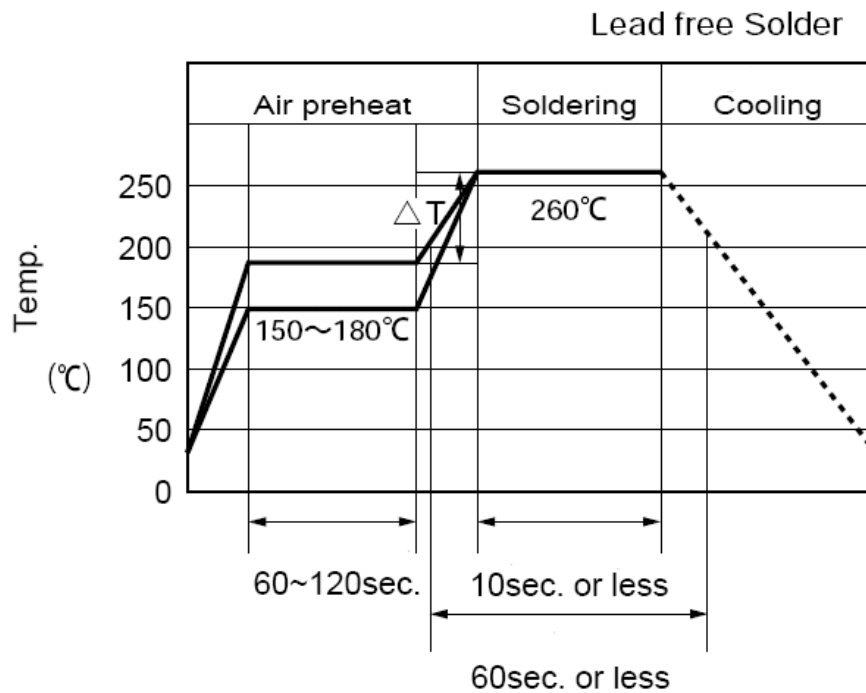
1000 pcs PA.11
 1 reel in small inner box
 Dimensions - 350*350*47



5 boxes / 5000 pcs in one carton
 Carton Dimensions - 370*360*275mm



8. Solder Reflow Profile



1. Time shown in the above figures is measured from the point when chip surface reaches temperature.
2. Temperature difference in high temperature part should be within 110°C.
3. After soldering, do not force cool, allow the parts to cool gradually.

General attention to soldering

- High soldering temperatures and long soldering times can cause leaching of the termination, decrease in adherence strength, and the change of characteristic may occur.
- For soldering, please refer to the soldering curves above. However, please keep exposure to temperatures exceeding 200°C to under 50 seconds.
- Please use a mild flux (containing less than 0.2wt% Cl). Also, if the flux is water soluble, be sure to wash thoroughly to remove any residue from the underside of components that could affect resistance.

Cleaning

When using ultrasonic cleaning, the board may resonate if the output power is too high. Since this vibration can cause cracking or a decrease in the adherence of the termination, we recommend that you use the conditions below.

Frequency: 40kHz
 Output Power: 20W
 Cleaning Time: 5 minutes max

Changelog for the datasheet

SPE-11-8-091 – PA.11

Revision: E (Current Version)

Date:	2022-12-06
Changes:	Full datasheet update.
Changes Made by:	Gary West

Previous Revisions

Revision: D

Date:	2022-05-12
Changes:	Updated Packaging Specifications
Changes Made by:	Paul Doyle

Revision: D

Date:	2020-11-10
Changes:	Updated to new format
Changes Made by:	Dan Cantwell

Revision: C

Date:	2016-01-06
Changes:	Added Packaging
Changes Made by:	Jack Conroy

Revision: B

Date:	2015-08-24
Changes:	Amended note on Gain
Changes Made by:	Aine Doyle

Revision: A (Original First Release)

Date:	2011-05-09
Notes:	
Author:	Technical Writer



www.taoglas.com

