
AT09023: 2.4GHz Printed Meander Antenna Reference Design

Atmel Wireless

Features

- Compact PCB antenna for 2.4GHz ISM band
- Return Loss, gain and efficiency measurement results
- Easy to integrate Altium® design files and Gerber files
- Guidelines to integrate antenna board with wireless board

Description

Scope of this application note is to guide customers to design applications using ATEVK-2400-ANT-M Printed Meander antenna for 2.4GHz ISM band based on AT86RF233 transceiver.

This application note contains the physical dimensions of the antenna, RF measurement results, and a brief design note. It describes the methods to integrate this Antenna with a 2.4GHz Wireless board for quick Prototyping.

Table of Contents

1	Introduction	3
2	Printed Meander Monopole Antenna.....	3
2.1	Antenna Structure	3
2.2	Return Loss Measurement.....	5
2.3	Radiation Pattern, Gain, and Efficiency Measurement.....	5
2.4	Guidelines for Antenna Integration with Transceiver.....	9
3	Abbreviations and Acronyms	9
4	References	9
5	Revision History	10

1 Introduction

This application note guides in designing a new 2.4GHz Printed Meander Antenna based on system requirements, board size, and layer stack up.

This Antenna Reference design can be integrated with any 2.4GHz ISM band transceiver to verify their prototypes quickly and it acts as a better reference for designing an application using antenna.

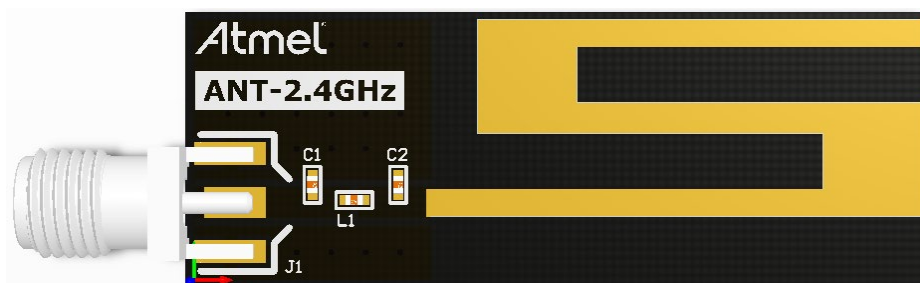
2 Printed Meander Monopole Antenna

2.1 Antenna Structure

The 2.4GHz ISM band (2.402GHz ~ 2.484GHz) is commonly used unlicensed band for Industrial, Scientific, and Medical Applications. Demand for compact antennas operating in this frequency band is increasing day by day. A standard $\lambda/4$ Printed Monopole Antenna (λ = wavelength) is widely used in many applications due to its small size and better radiation characteristics. The operation of Monopole is similar to dipole. The ground plane of the Monopole acts as the second arm to perform a Dipole operation [1]. Any modifications to the size of the ground plane, affects the performance of antenna. So, the designer must be careful about the ground plane size during the design and it must be fixed to the actual board size of the application.

Using Meander structure, the size of the monopole can be reduced to fit the available board size with a small degradation in antenna efficiency and radiation characteristics when compared with standard antenna. Meander structure actually requires longer trace (in multiple hair-pin bends) than $\lambda/4$ for its operation. However, it occupies lesser space. The structure of the proposed Printed Meander is shown in Figure 2-1 and the dimension of the antenna is shown in Figure 2-2.

Figure 2-1. 2.4GHz ISM Band Printed Meander Antenna with SMA Connector



Using Agilent's ADS tool Return Loss, Gain, Directivity, Efficiency, and Far-Filed Pattern Cuts were tuned in simulation before creating the prototype. PCB parameters are specified in Table 2-1 PCB Layer stack up is shown in Figure 2-3. Feed line is a 50 Ω "CPW with Ground Plane" structure with trace width W=1.6mm and Trace-Ground gap of 0.4mm on either side. An edge mount SMA connector foot print is placed in the board to assemble SMA connector for measurement and integration purposes.

Table 2-1. PCB Characteristics

PCB Material – Parameter	Value
PCB substrate	FR-4
Dielectric constant (ϵ_r)	4.4
Loss tangent ($\tan\delta$)	0.02
Substrate thickness	1.6mm
Cu thickness	35 μ m

Figure 2-2. Dimension of 2.4GHz ISM Band Printed Meander Antenna

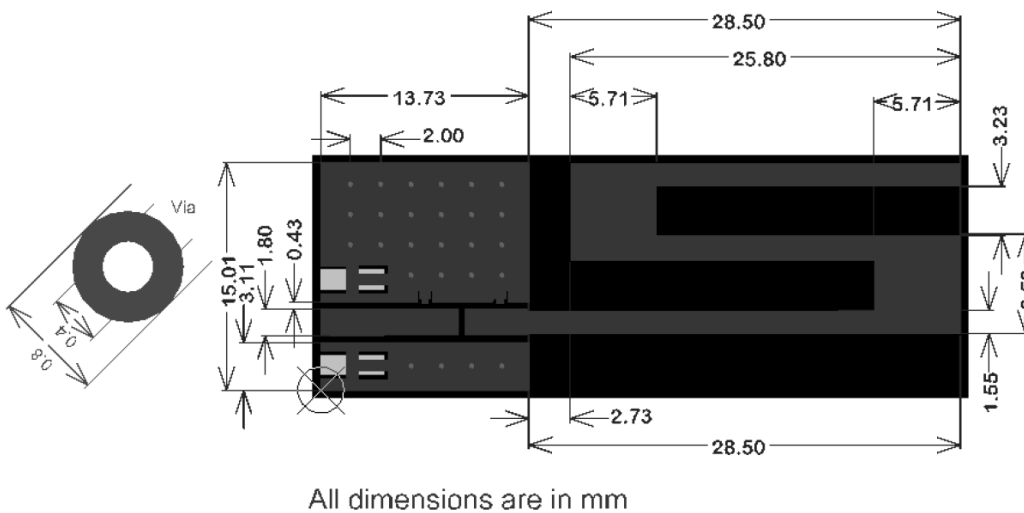
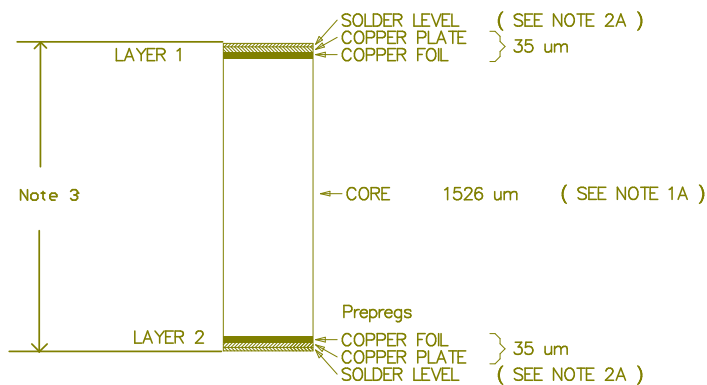


Figure 2-3. PCB Layer Stackup

Layer stackup



If a standard printed monopole is used for the PCB layer stack up, it requires a $\lambda_g/4$ length of 17mm excluding the ground plane size. The length of the ground plane must be $\geq \lambda_g/4$ for standard monopole.

$$\lambda_g = \frac{C}{\sqrt{\epsilon_r}f}$$

Where,

- λ_g = Guided wavelength
- C = Velocity of light
- f = Frequency of operation
- ϵ_r = Relative permittivity of dielectric material

The size of the Printed Meander antenna including ground plane is 42.23mm x 15.01mm.

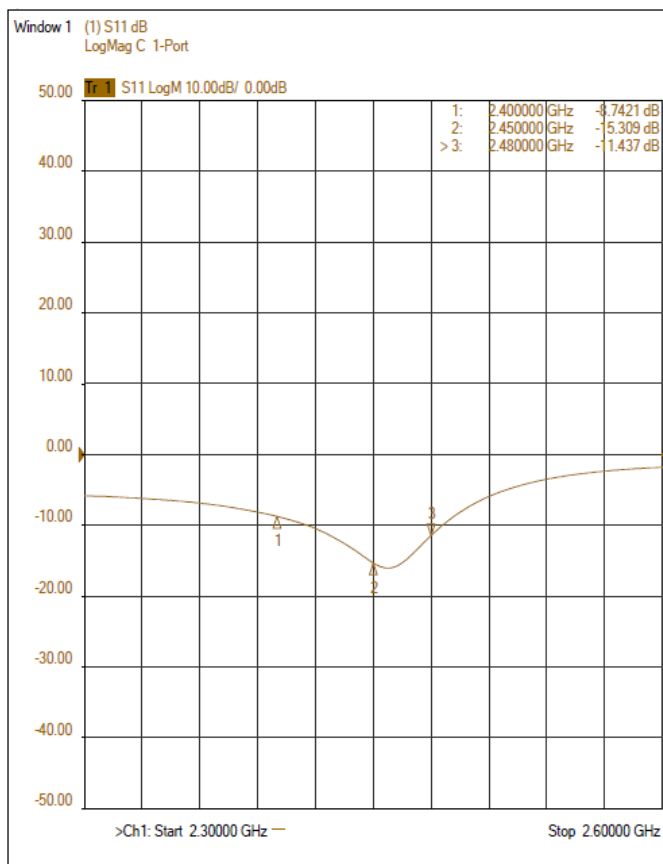
2.2 Return Loss Measurement

Return Loss (S11) is directly measured in a Calibrated Vector Network Analyzer. The measured Return Loss is shown in [Figure 2-4](#). The antenna was tuned to operate in the desired band with a series Inductor of $L1 = 8.2\text{nH}$ and it achieved Return Loss $< -8.74\text{dB}$ for $2.4\text{GHz} \sim 2.5\text{GHz}$ band.

The antenna can also be tuned to operate in the desired band in the prototype board based on the following changes.

The resonant frequency of the antenna can be increased by reducing the length (trace perpendicular to ground plane) of the trace and it can be decreased by increasing length of the trace. Length can be increased using copper foil. Bandwidth can be improved by increasing the width of the line.

Figure 2-4. Measured Return Loss Plot



2.3 Radiation Pattern, Gain, and Efficiency Measurement

[Figure 2-5](#), [Figure 2-6](#), and [Figure 2-7](#) show Azimuth and Elevation pattern-cuts of the antenna. These patterns are not normalized with respect to peak value. Antenna was configured in receive mode for all these measurements. The plots represent measured power density by placing the Receiving antenna in different orientations. From the results, the pattern is fairly Omni-directional in both Azimuth and Elevation planes.

Figure 2-5. Azimuth Radiation Pattern ($\theta=90^\circ$)

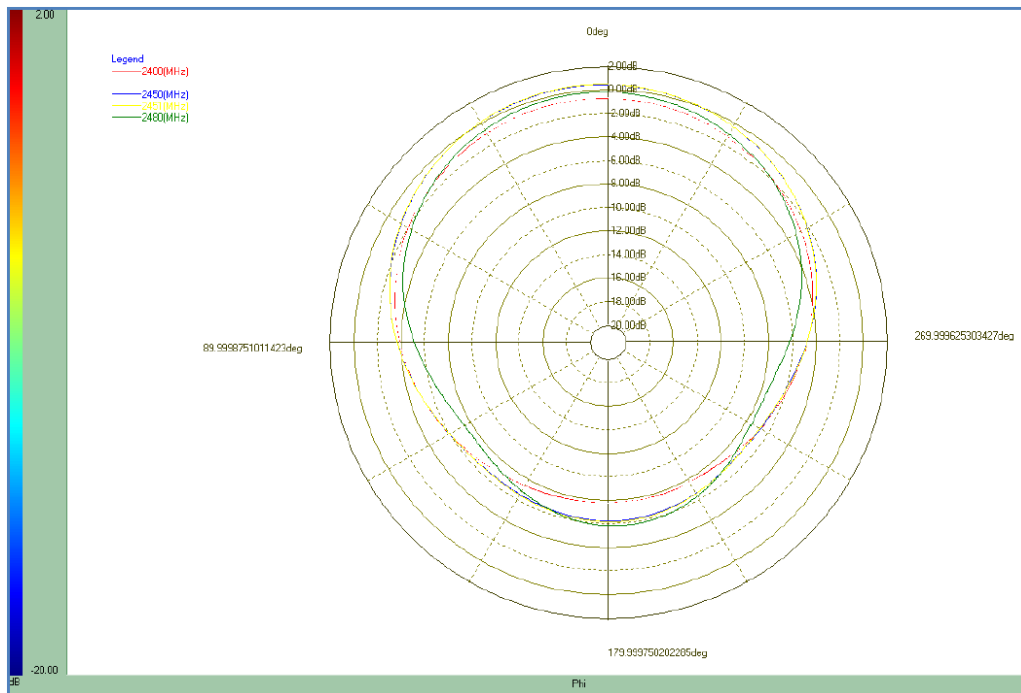


Figure 2-6. Elevation Radiation Pattern ($\phi=0^\circ$)

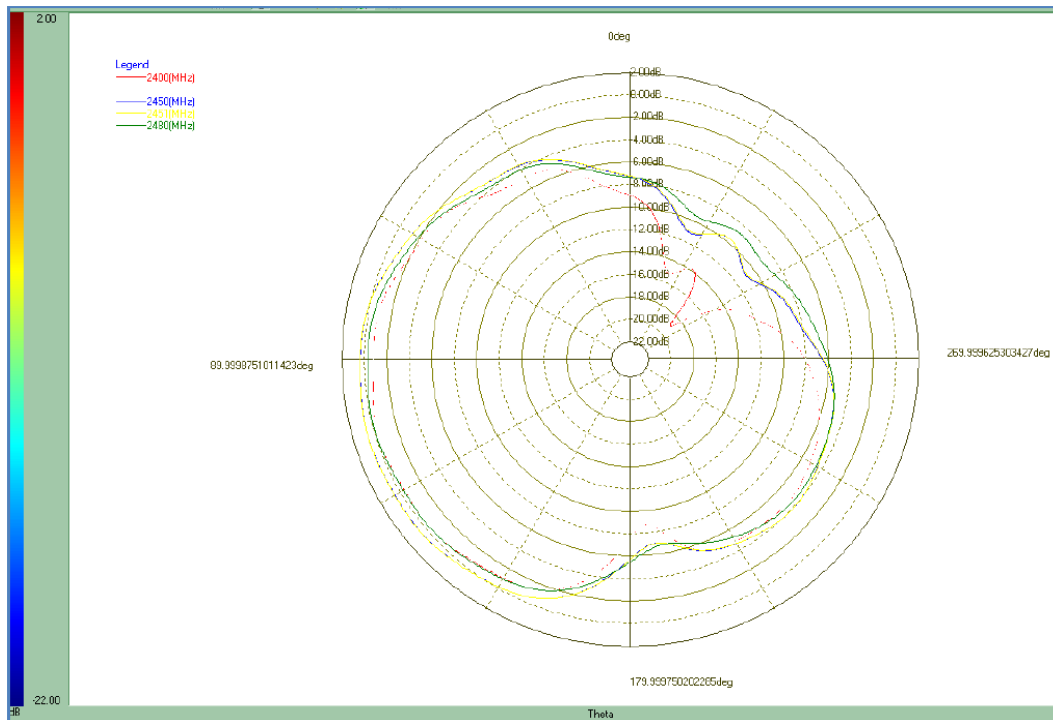


Figure 2-7. Elevation Radiation Pattern ($\phi=90^\circ$)

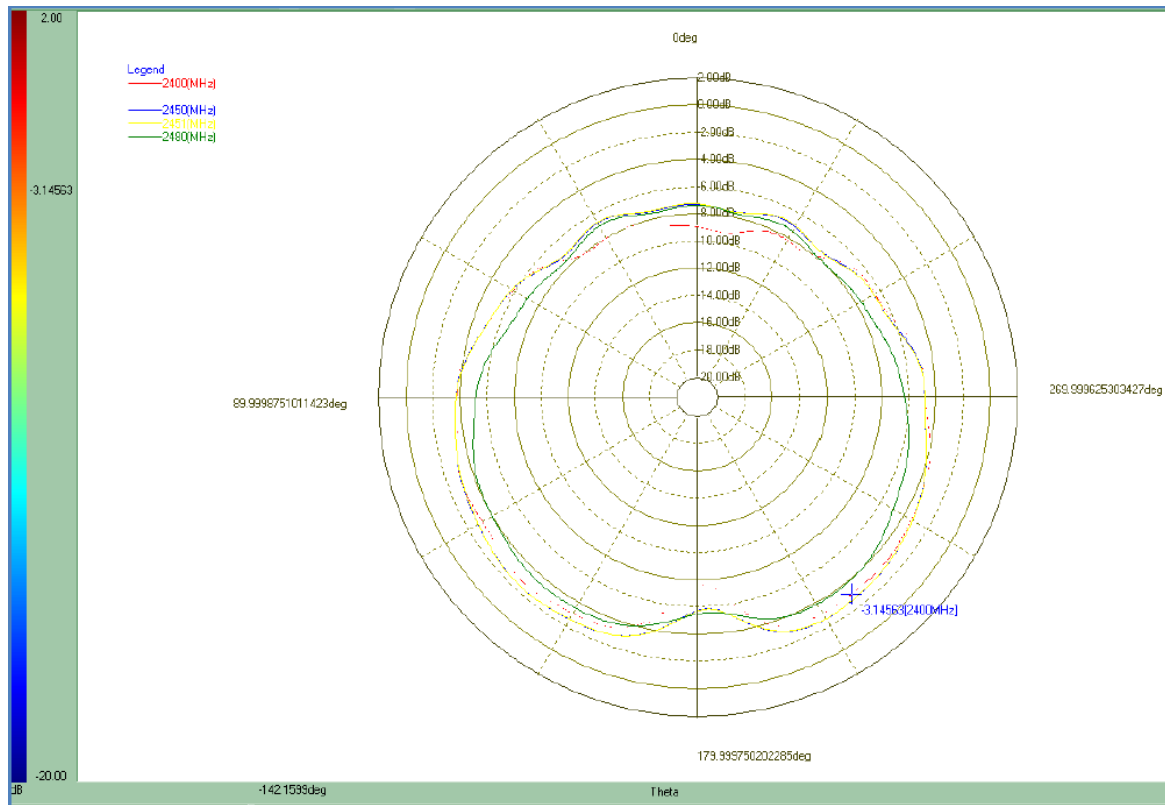


Figure 2-8. 3D – Radiation Pattern: Front View

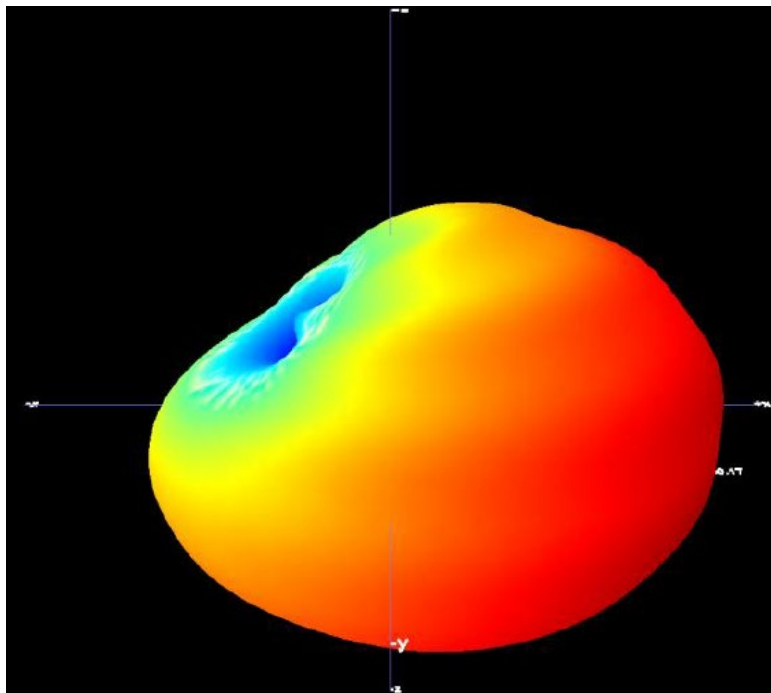


Figure 2-9. 3D – Radiation Pattern: Top View

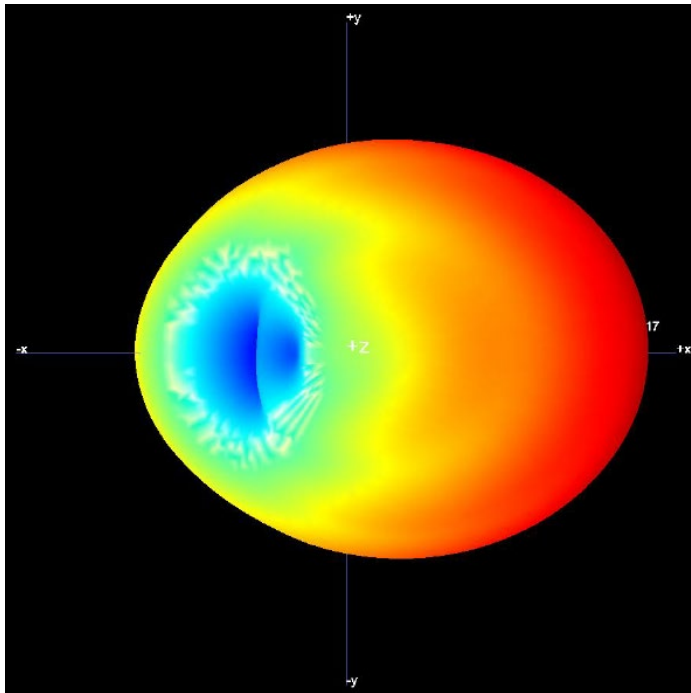
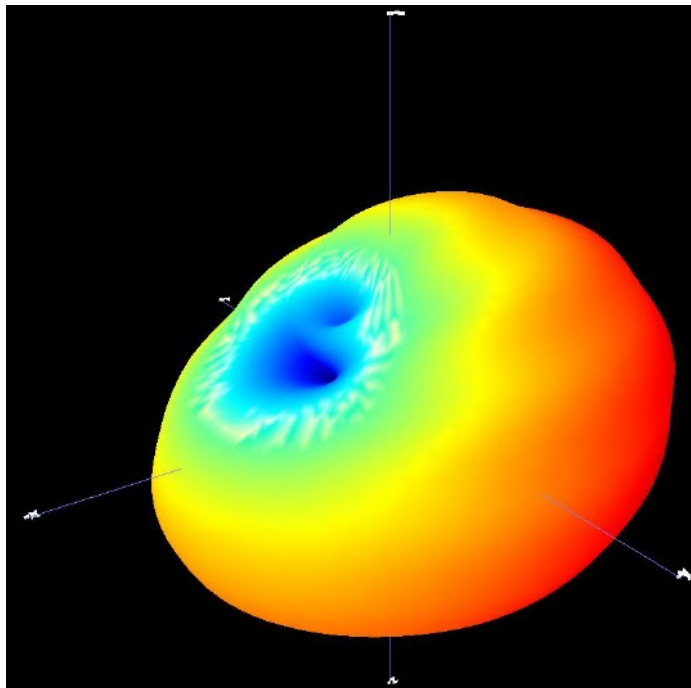


Figure 2-10. 3D – Radiation Pattern: Side View



The measured peak gain of the antenna is 0.67dB and the efficiency is 46.23%. Usually Efficiency of the PCB antennas is based on antenna type, and ground plane size. This reference design has a very small ground plane and hence the efficiency is slightly less. If the ground plane is large enough in the final application board, the efficiency might be higher.

2.4 Guidelines for Antenna Integration with Transceiver

- Although, the discussed reference board contains only two layers, this reference design can be used for multilayer application. In multilayer application, antenna can be placed either on top layer or bottom layer based on the available space. Any change in Dielectric constant or tangent loss of the material or thickness of the board will shift the resonant frequency and change radiation characteristics. These factors must be considered while designing the application.
- It is usually a better practice to place the transceiver and antenna on the same layer without having any via-hole in RF path. Via holes create discontinuity and loss of signal which will require proper impedance matching to reduce loss.
- Copper clearance must be provided in all layers beneath the antenna. Because, conductor will reflect RF signal and change the radiation characteristics of the antenna.
- RF trace connecting Transceiver output (Balun output for Differential output Transceivers) and Antenna Feed point must be 50Ω controlled impedance line. It is highly preferred to have a provision for Pi-pad matching network to improve matching between Transceiver output and antenna input. It will avoid board re-spin during mismatched circumstances. Many combinations of lumped element matching methods are available [2].
- Mechanical enclosure of the product must not have any conductive material. Plastic enclosure is preferred; but, it might also detune the antenna from the desired band. So, tuning the antenna with lumped components must be performed by placing it within the enclosure.
- RF and antenna board can be placed separately. If space is not a constraint, SMA cables can be used to connect wireless board with antenna board. Insertion loss of a standard 20cm SMA-Male to SMA-Male cable varies from 0.15dB ~ 0.5dB at 2.45GHz (depending on the manufacturer). Most ISM band applications are designed on small sized boards. A small board mount RF connector such as U.FL and mating flexible RF cables can be used to connect RF board with Antenna board.

3 Abbreviations and Acronyms

BW	:	Bandwidth
CPW	:	Coplanar Waveguide
GND	:	Ground
GHz	:	Giga Hertz
ISM	:	Industrial, Scientific and Medical
MHz	:	Mega Hertz
mm	:	Millimeter
PCB	:	Printed Circuit Board
RF	:	Radio Frequency
Rx	:	Receiver
SMA	:	Sub-Miniature Version A
Tx	:	Transmitter
μm	:	micro-meter

4 References

- [1] “Antenna Theory: Analysis and Design”, Third Edition, Constantine A. Balanis
- [2] “Microwave Engineering”, Fourth Edition, David M.Pozar

5 Revision History

Doc Rev.	Date	Comments
42306A	07/2014	Initial document release.

