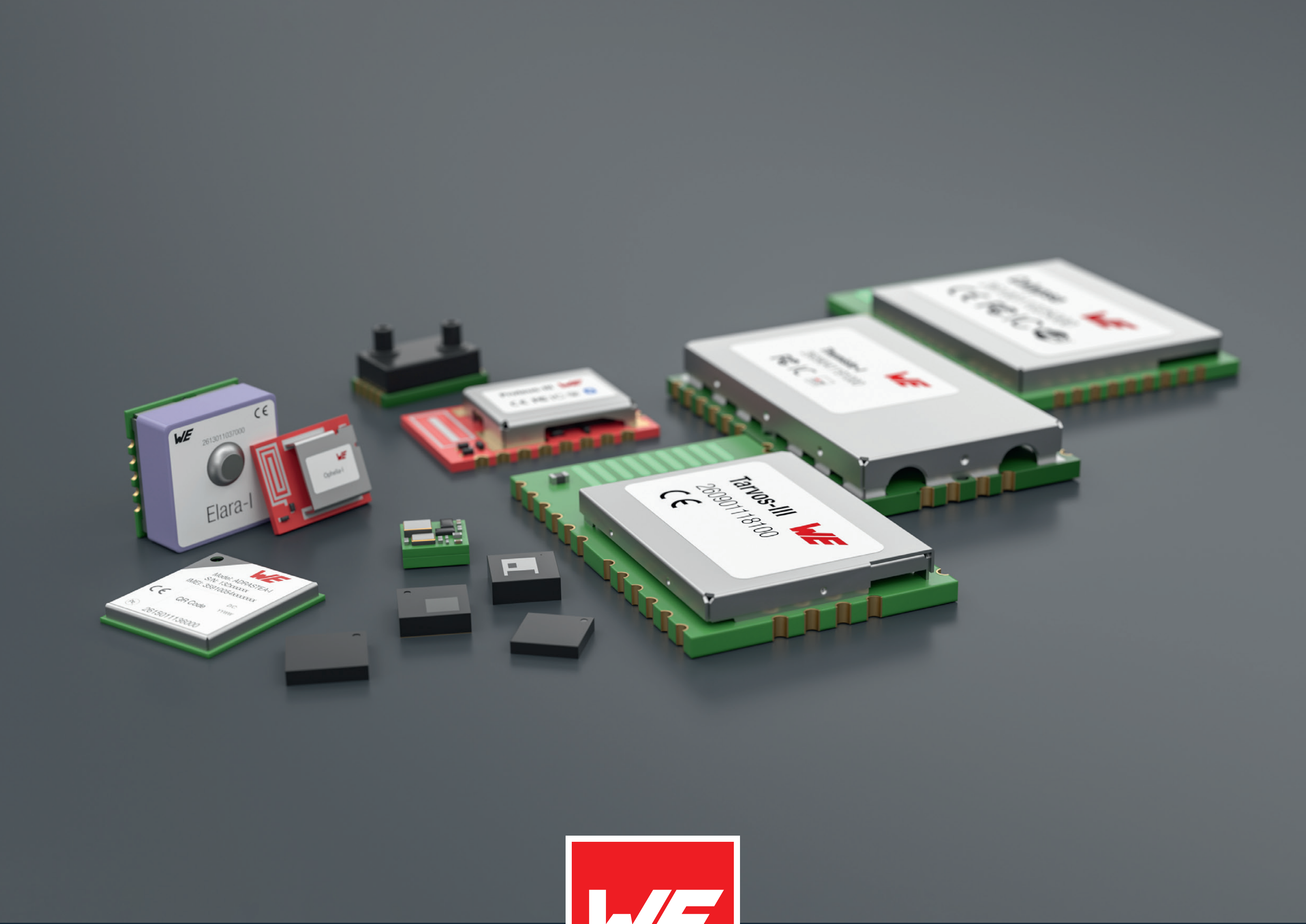

ANR033

Ground plane effects on radio module antennas

Version 1.0

July 3, 2025

WÜRTH ELEKTRONIK MORE THAN YOU EXPECT

Revision history

Manual version	Notes	Date
1.0	Initial Version	July 2025

Abbreviations

Abbreviation	Description
3D	3-Dimensional
AR	Axial Ratio
CP	Circular Polarization
FR4	Flame Retardant 4
IFA	Inverted-F Antenna
LNA	Low Noise Amplifier
MIFA	Meandered Inverted-F Antenna
PIFA	Planar inverted F-antenna
PCB	Printed Circuit Board
RF	Radio frequency
RHCP	Right Hand Circular Polarization
SAW	Surface Acoustic Wave
SMD	Surface Mounted Device
THT	Through Hole Technology
TM	Transverse Magnetic
VSWR	Voltage Standing Wave Ratio
Wi-Fi	Wireless Fidelity

Contents

1 Introduction

In recent years, the demand for wireless communication has increased exponentially, and it has become crucial to design small, low-cost, and efficient antennas. The Printed Circuit Board (PCB) antenna is a popular choice for wireless communication applications due to its low profile, easy integration with the circuitry, and cost-effectiveness. PCB antennas can be designed to work in a variety of frequency bands and can be placed on different areas of the PCB. Some of the important aspects of designing a PCB antenna is to understand its resonant frequency and radiation pattern. The characteristics of an antenna can be influenced by several external factors including the antenna itself. One factor that can significantly affect the characteristics of PCB antennas is the ground plane.

This application note explores common antenna types, designs, characteristics and the effects of the ground plane on radio module PCB antennas.

2 Antenna types and designs

An antenna can be described as a device used to radiate and receive electromagnetic waves. It transforms the electromagnetic waves from the free space into electrical voltage and current in a conductor structure and vice versa. The antenna is an essential component in any radio communication system.

The antenna type and design is a critical aspect of modern electronics. With the increasing demand for wireless connectivity, the importance of antenna design cannot be overstated. Antenna design involves the creation of a complete system that includes the antenna, matching circuit, and the feed line. The goal of the antenna design is to maximize the power transfer between the transmitter and receiver while minimizing the losses.

2.1 Different types of antennas

There are various types of antenna designs, each with its unique characteristics and applications. Some of the most common types of antennas:

2.1.1 Monopole antennas

A monopole antenna is the most basic type of an antenna. It consists of a single wire element that is perpendicular to the ground plane. The length of the wire is typically a quarter wavelength, which is determined by the frequency of operation.

The monopole antenna can be placed in the center of a PCB or at one end, depending on the desired radiation pattern. It consists of a radiating element on one side of the PCB and a ground plane on the other side. The ground plane acts as the second radiating element. The single radiating element along with the ground plane acts as complete dipole antenna. Due to this, monopole antennas are highly dependent on their ground plane. A monopole antenna has a typical impedance of around 36 Ω when operated close to an ideal ground reference and therefore has an asymmetrical port. The monopole antenna in its classic shape has an omni-directional radiation pattern. By changing the shape and groundplane it is possible to adapt the shape of the radiation pattern.

2.1.2 Dipole antennas

A dipole antenna is another commonly used PCB antenna. It consists of two wire elements that are identical to each other and aligned horizontally or vertically to the ground plane. The length of each element is typically a half wavelength, which is determined by the frequency of radio operation.

Similar to the monopole antenna, the dipole antenna in its classic shape has an omni-directional radiation pattern. The impedance of a half-wavelength dipole is around 73 Ω. Dipole antennas do not require an ground reference as they are symmetrical.

2.2 Common antenna designs for PCB antennas

PCB antennas are becoming increasingly popular due to their compact size and low cost. The antenna types mentioned above in the section 2.1 can also be realized on the PCB. However, due to miniaturization, most PCB antennas are commonly designed as monopole antennas. In a monopole antenna, the ground plane serves as a ground reference, not as a second radiating element. The monopole itself is a single radiating element, typically a quarter-wavelength long. The ground plane (or an actual ground surface) reflects the electromagnetic waves, effectively creating a virtual image of the monopole that simulates the second half of a dipole antenna. Due to this, the antenna performance can be enhanced or reduced by modifying both, the antenna and the ground plane. Some of the most common antenna designs for such PCB antennas are given in the following sections:

2.2.1 Inverted-F antenna

An Inverted-F Antenna (IFA) is a type of antenna commonly used in wireless communication systems. It gets its name from its distinctive shape, which resembles the letter F when viewed from the side. The Inverted F-Antenna typically consists of a vertical radiating element that is bent at a right angle, forming the F shape.

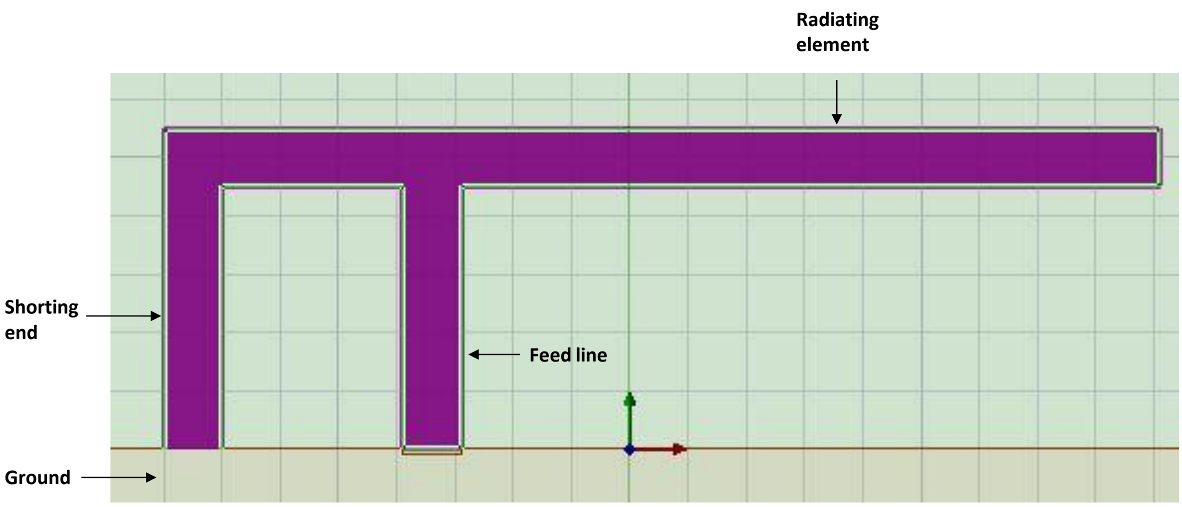

Figure 1: Design pattern of an inverted F-antenna

The antenna consists of a conductive plate or ground plane, a feed line or transmission line, and a radiating element, which is a conductive strip that is connected to the feed line at one end and shorted to the ground plane at the other end. The feed line is typically a microstrip or coplanar transmission line, and the radiating element is typically parallel to the ground plane, with its shorted end close to the ground plane.

The IFA's operating frequency is determined by the length of the radiating element, which is typically one-quarter of the wavelength of the signal. The ground plane beneath the antenna acts as a reflector and enhances the antenna's radiation efficiency, while the shorted end of the radiating element acts as a tuning stub, which provides impedance matching between the feed line and the antenna.

The directivity of an Inverted F-Antenna typically ranges from 2 to 6 dBi, depending on its design and orientation, indicating how well it focuses energy compared to an isotropic radiator. Its gain usually falls between 3 to 6 dBi, reflecting its effectiveness in directing radio frequency energy. The input impedance is generally designed to work around 50 or 75 Ω, crucial for matching with standard transmission lines to minimize reflections and maximize power transfer. The bandwidth often varies from 10% to 20% of the center frequency, with some designs achieving broader bandwidths. The radiation pattern is generally omnidirectional in the horizontal plane, with a more focused pattern in the vertical plane, making it suitable for various wireless communication applications.

IFAs are widely used in mobile devices like smart phones, tablets, and laptops due to their compact size and ease of integration. They can also be designed to operate on multiple frequency bands, making them suitable for use in multiband communication systems. These antennas are commonly used in Bluetooth® and WiFi applications.

2.2.2 Meandered antenna

Meandered antennas are a type of radio frequency antenna that are designed with a series of bends or folds in the radiating element. The meandering shape allows for a larger physical length of the antenna, which can lead to improved performance in terms of resonance and bandwidth, without increasing the overall size of the antenna. This size reduction is beneficial for small form factor applications such as mobile devices, wireless sensors, and other compact electronics.

The meandering pattern can take many forms: zig-zags, spirals, or loops. By adjusting the length and spacing of the meanders, the antenna is tuned to operate at specific frequencies or frequency ranges. This provides the possibility to adjust the antenna design parameters to achieve better performance. Meandered antennas can be designed to have directional or omni-directional radiation patterns based on the groundplane its designed with.

Figure 2: Design pattern of a meandered PIFA antenna

One major advantage of meandered antennas is their ability to be easily integrated into electronic devices, such as printed circuit boards, which can simplify manufacturing and reduce costs. They are also less sensitive to orientation compared to other antenna designs, which can be beneficial for applications where the device may be moved or rotated.

However, meandered antennas can be more complex to design and manufacture than other antenna types, and their performance can be affected by nearby components and structures. As with any antenna design, careful consideration must be given to the specific application requirements and environmental factors to optimize the performance of a meandered antenna. These antennas are commonly used in 2.4 GHz applications. Most of the PCB antennas are a form of meandered antenna, due to the need of miniaturization in size.

The directivity of a meandered PIFA typically ranges from 2 to 5 dBi. This value varies based on the specific design and frequency of operation. The gain usually falls between 2 to 4 dBi, indicating its ability to direct energy effectively. The input impedance is generally designed to work around 50 or 75 Ω, facilitating compatibility with standard transmission lines. The bandwidth can vary but is often around 10% to 20% of the center frequency.

Some designs may achieve broader bandwidths, enhancing versatility for various applications. The dimensions of a meandered PIFA can be quite compact, often around 30 mm x 10 mm or smaller, depending on the design and target frequency. The radiation pattern is typically omnidirectional in the horizontal plane, with some directivity in the vertical plane, making it effective for mobile communications. Efficiency values are usually in the range of 70% to 90% depending on the design and operating conditions.

2.2.3 Helical PCB antenna

Helical antennas are compact, high-gain PCB antennas featuring a spiral trace wound around a central axis. The length and diameter of the helix are determined by the operating frequency, typically ranging from half to one wavelength of that frequency. Depending on the number of turns and spacing, these antennas can be designed for either a directional or omnidirectional radiation pattern, with single-turn helices providing omnidirectional coverage and multi-turn designs enhancing gain by focusing energy directionally.

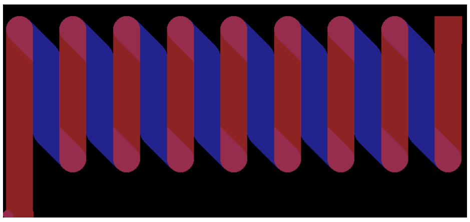

Figure 3: Design pattern of a helical PCB antenna.

Helical antennas can support either linear or circular polarization, the latter being advantageous for applications with varying antenna orientations, such as satellite communications. The input impedance is generally designed to work around 50 or 75 Ω, ensuring compatibility with standard transmission lines for efficient power transfer. Gain typically ranges from 8 to 20 dBi, influenced by the number of turns and overall design. These antennas offer a moderate bandwidth, usually around 10% to 20% of the center frequency, with multi-turn designs providing broader bandwidths due to their higher quality factor. Their compact size makes them suitable for various devices, with efficient designs often exceeding 80% efficiency.

Helical antennas are widely used in applications like satellite communications, GPS, telemetry, and wireless data transmission, where their size, gain, and polarization versatility are particularly beneficial.

The helical PCB antenna is a form of meandered antenna and is possible to design it linear or circular polarization. Based on the antenna design parameters of the helical antenna the performance of the antenna varies.

2.2.4 Patch antennas

Patch antennas are popular in wireless communication due to their low profile and ease of fabrication. They consist of a radiating patch element on one side of the PCB and a ground plane on the opposite side, with the patch available in square, rectangular, or circular shapes, each affecting performance. Typically designed for a directional radiation pattern, patch antennas are suitable for focused signal transmission and may support circular polarization for improved performance in varying orientations.

Directivity ranges from 6 to 12 dBi, with higher values indicating stronger energy focus. The input impedance is generally designed to work around 50 or 75 Ω, ensuring compatibility with standard transmission lines for effective power transfer. Patch antennas have a narrow bandwidth of about 5% to 10% of the center frequency, which can be enhanced through techniques like using thicker substrates or multi-layer designs. Efficiency typically exceeds 70%, influenced by factors such as surface and dielectric losses.

The dimensions are often a fraction of the operating wavelength, allowing for compact integration in devices. These antennas are commonly used in applications like GPS, mobile communications, and Wi-Fi systems, where their compact size and directional capabilities are advantageous.

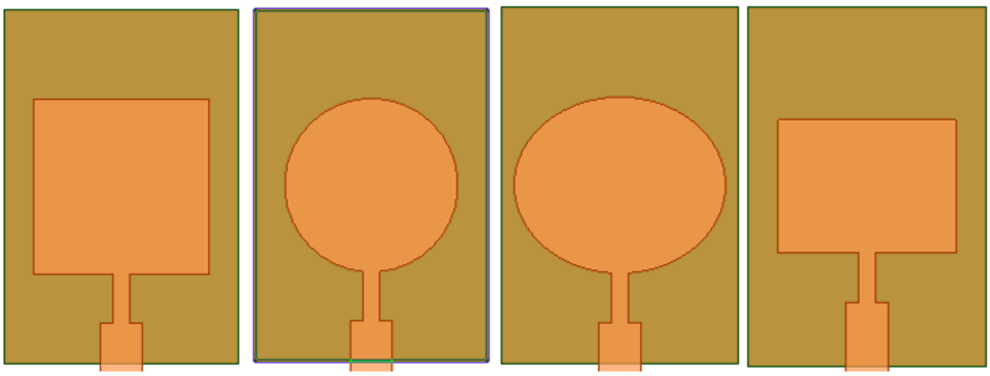

Figure 4: Design pattern of different patch antennas:Square, Circular, Oblique, Rectangular

3 Antenna characteristic parameters

The performance of an antenna depends on several factors such as the antenna type, shape, size, and the frequency of operation. To understand how the antenna performance varies, understanding the antenna characteristics is absolutely necessary. The radiation performance of antenna can be described through some important antenna characteristics as follows:

3.1 Resonant frequency

Resonant frequency is the frequency at which the impedance of the antenna is entirely real. The imaginary impedance is zero as the capacitive and inductive impedances cancel each other out. The resonant frequency is determined by the physical dimensions of the antenna and the dielectric constant of the PCB substrate.

The resonant frequency of a PCB dipole antenna can be calculated using the following formula:

$$\begin{aligned}
f &= \frac{c}{2L_{eff} \times \sqrt{\epsilon_{eff}}} \\
\epsilon_{eff} &= \frac{\epsilon_r + 1}{2} + \frac{\epsilon_r - 1}{2} \times (1 + 12h/W)^{-0.5} \\
L_{eff} &= L + 2\Delta L \\
\Delta L &= -0.412h \times (\epsilon_{eff} + 0.3) \times (W/h + 0.264)/(\epsilon_{eff} - 0.258) \times (W/h + 0.8)
\end{aligned}$$

f is the resonant frequency [Hz]
c is the speed of light in vacuum (3×10^8 [m/s])
L is the length of the antenna [mm]
ϵ_r is the relative dielectric constant of the PCB substrate
W is the width of the antenna [mm]
ϵ_{eff} effective dielectric constant
L_{eff} is the thickness of the substrate [mm]
h is the relative dielectric constant of the PCB substrate [mm]

As the length of the antenna increases, the resonant frequency decreases. Similarly, as the dielectric constant of the PCB substrate increases, the resonant frequency decreases. To design a PCB antenna with a desired resonant frequency, the physical dimensions of the antenna and the dielectric constant of the PCB substrate must be carefully chosen.

3.2 Bandwidth

The bandwidth of a PCB antenna is a critical parameter that indicates the range of frequencies over which the antenna can operate efficiently. It is typically defined as the frequency range where the antenna meets specific performance criteria, such as a minimum level of gain or efficiency. Bandwidth can range from a few megahertz (MHz) to several gigahertz (GHz), depending on the antenna design and application.

A wider bandwidth is often desirable as it allows the antenna to function effectively across

multiple frequency bands, making it suitable for various applications, including wireless communication systems (e.g., cellular, Wi-Fi, and Bluetooth®).While wideband antennas can accommodate a range of frequencies, they may also inadvertently pick up unwanted signals from other sources. This can negatively impact the antenna's electromagnetic immunity and overall performance. Antennas designed for specific frequency ranges are essential to minimize interference and overlap with other technologies, ensuring reliable communication.
Bandwidth Calculation: The bandwidth of an antenna can be calculated using the following formula:

$$BW = \frac{f_{high} - f_{low}}{f_{center}}$$

$$f_{center} = \frac{f_{high} + f_{low}}{2}$$

BW is Bandwidth [Hz]
f_{high} is upper frequency limit of the operational range [Hz]
f_{low} is lower frequency limit of the operational range [Hz]
f_{center} is center frequency [Hz]

PCB antennas can exhibit varying bandwidths:
Narrowband antennas: <5% of the center frequency.
Wideband antennas: 10% to 30% or more of the center frequency

Techniques for Bandwidth Enhancement:
Increasing the width of the radiating element. Employing notched structures or slots in the design. Using multi-layer PCBs to reduce dielectric effects.

Trade-offs: While a wider bandwidth is beneficial, it often comes at the cost of gain and directivity, as broadening the bandwidth can lead to reduced performance in specific frequency ranges.

3.3 Radiation pattern

The radiation pattern of a PCB antenna is a graphical representation of the antenna radiating electromagnetic waves in space. It can be represented in both 2D and 3D plots, showcasing the power of the radiated wave in various directions. Based on design and implementation, the radiation pattern of a PCB antenna can be omni-directional or directional.

An omni-directional radiation pattern means, that the antenna radiates electromagnetic waves equally in all directions on one plane with minimum radiation in the direction perpendicular to this plane. This shape fits to a lot of applications for example on one floor of a building, or on the surface of the earth, where the position of the devices is not fixed. So Omni-directional antennas are commonly used for applications where the antenna needs to communicate with devices in multiple directions, such as in a WiFi router.

A directional radiation pattern means that the antenna radiates electromagnetic waves in a specific direction or a narrow beam. Directional antennas are commonly used for applications where the antenna needs to communicate with devices in a specific direction or in a long-range communication link.

Considering the radiation pattern of PCB antennas is crucial for optimizing their performance in specific applications. Whether omnidirectional for widespread coverage or directional for targeted communication, the choice of radiation pattern affects the antenna's effectiveness. By considering parameters such as directivity, gain, radiation efficiency, and polarization, designers can tailor PCB antennas to meet the demands of various wireless communication scenarios.

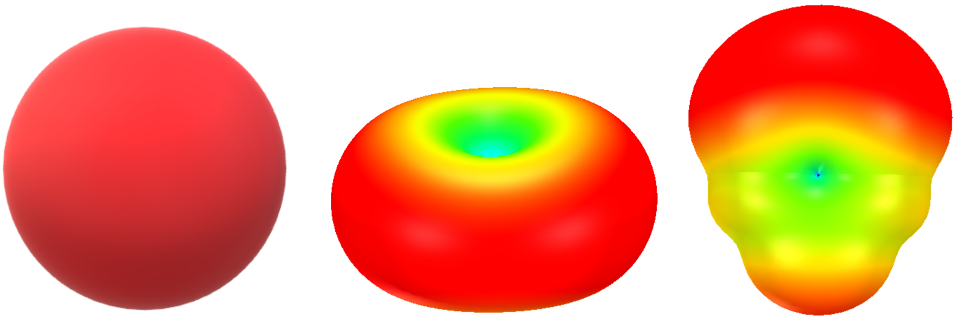

Figure 5: Isotropic, Omni-directional and Unidirectional radiation pattern

3.4 Efficiency

The efficiency of a PCB antenna is a measure of how much of the power supplied to the antenna is actually radiated as electromagnetic waves. In other words, it's a measure of how well the antenna converts electrical power into radio waves. A more efficient antenna will be able to radiate more power and receive more signal power, resulting in better performance.

$$\eta_{rad} = \frac{P_{rad}}{P_{in}}$$
$$P_{in} = P_{rad} + P_l$$

P_{rad} it the radiated power
P_{in} is the input power accepted by the antenna
P_l is the power loss
η_{rad} is the radiation efficiency

The efficiency of a PCB antenna can vary from a few percent to over 90%. The total efficiency takes the power losses as well as the effect of impedance matching into account. The total Efficiency η_{total} takes into account all losses, including resistive losses and those due to impedance mismatch. Both total and radiation efficiency can be used to express antenna gain.

$$\eta_{total} = \frac{\eta_{rad}}{\eta_{match}}$$

η_{total} is the total efficiency
η_{match} is the efficiency of impedance matching

3.5 Directivity

The directivity of a PCB antenna is a measure of the antenna's ability to focus electromagnetic waves in a specific direction. It's a ratio of the radiation intensity in a specific direction to the average radiation intensity in all directions.

Directivity is often confused with gain, but they are not the same. Gain is a measure of the power radiated by the antenna in a specific direction, while directivity is a measure of the antenna's ability to focus radiation in that direction. An antenna with high directivity will have a narrow beam-width, which means that it will radiate most of its power in a specific direction. An antenna with low directivity will have a wide beam-width, which means that it will radiate its power more evenly in all directions.

It is defined as the ratio of the radiation intensity U in a given direction θ to the average radiation intensity Uavg radiated in all directions. The directivity D can be mathematically expressed as:

$$D = \frac{U\theta}{U_{avg}}$$

D is Directivity (dimensionless)
$U\theta$ is the radiation intensity in the desired direction
$Uavg$ is the average radiation intensity over all directions

The directivity of a PCB antenna can be influenced by several design parameters, including Antenna Shape (geometry), size and the feeding mechanism. Larger antennas can achieve higher directivity and the method of feeding the antenna can alter its radiation characteristics.

3.6 Gain

The antenna gain refers to the measure of power radiated by the antenna in the direction of maximum radiation compared to its average referred to the supplied power which is typically referred to a known radiator.

Gain is expressed in decibels (dB) and can be specified in two common forms:

Isotropic Gain (dBi): This measures the gain relative to an ideal isotropic radiator, which radiates power uniformly in all directions. The gain in this context is denoted as dBi.

Dipole Gain (dBd): This measures the gain relative to a half-wave dipole antenna. It is calculated by adding 2.15 dB to the dBi value because a half-wave dipole radiates about 2.15 dB more power than an isotropic radiator. Antenna gain is a function of the directivity of the antenna and its efficiency.

$$G = D \times \eta$$

G is the antenna gain
D is the directivity
η is the radiation efficiency

Antenna gain depends on two key factors: directivity and efficiency. Directivity measures an antenna's ability to focus energy in a specific direction, while efficiency indicates how well the antenna converts input power into radiated power.

There are two main types of gain:
Directional Gain: Found in antennas like Yagi or parabolic designs, which are optimized for specific communication links and suitable for long-range applications.

Omnidirectional Gain: Exhibited by monopole or dipole antennas, which radiate power equally in all directions and are ideal for broad coverage. Higher gain antennas enhance signal transmission and reception over longer distances, improving communication range and quality. However, they often require precise alignment compared to lower gain, omnidirectional antennas.

For PCB antennas, gain can vary significantly, typically ranging from a few dBi to several tens of dBi. The gain of microstrip antennas generally ranges from 0 to 10 dBi or more, depending on their size and shape. Specific gain values include approximately 3.15 dBi for a quarter-wave monopole antenna over an infinite ground plane and around 2.15 dBi for a half-wave dipole antenna.

3.7 Input impedance

The antenna input impedance refers to the impedance seen by the antenna at its input terminals. It is a complex quantity that includes both the resistance and reactance of the antenna. The input impedance of an antenna is important because it affects the amount of power that is transferred between the antenna and the transmission line.

$$Z_{in} = R_{in} + X_{in}$$

Z_{in} is the input impedance of the antenna
R_{in} is the input resistance
X_{in} is the capacitive or inductive reactance

The input impedance of an antenna is typically designed around 50 Ω, which is the standard impedance used in most radio systems.

The monopole antenna and the dipole antenna have different input impedances depending on their design and frequency of operation. A ideal quarter-wave monopole antenna over infinite ground plane has an input impedance of around 36 Ω, while a equivalent half-wave dipole antenna has an input impedance of around 75 Ω.

The most common impedance values are 50 Ω and 75 Ω. These values are initially originated from coaxial cables and analysis based on it. The analysis was made based three criterias voltage maximum, power handling capacity, and attenuation. The choice of impedance selection involves balancing tradeoffs. The 50 Ω value is a good compromise for power and voltage, so 50 Ω is most often used for radio transmitting and receiving applications. In contrast, for situations where low attenuation is the primary goal, such as with low-level signals from an antenna or an analog video link, 75 Ω is a better choice. So 75 Ω is primary used for video and audio systems.

It's important to match the input impedance of the PCB antenna to the impedance of the transmission line and the RF system to avoid signal reflections and losses. Impedance matching can be achieved by using matching networks such as baluns or by adjusting the dimensions of the antenna to achieve the desired impedance. The reflection coefficient, return loss and VSWR are different representations of how well the antenna is matched to the tranmission line to achieve maximum performance out of the antenna.

3.8 Reflection coefficient

The reflection coefficient of an antenna is a measure of how much power is reflected back from the antenna to the transmission line or system it is connected to. It is defined as the ratio of the amplitude of the reflected wave to the amplitude of the incident wave, expressed as a complex number.
The reflection coefficient can be calculated using the formula:

$$\Gamma = \frac{(Z_{in} - Z_{out})}{(Z_{in} + Z_{out})}$$

Γ is the reflection coefficient
Z_{in} is the input impedance of the antenna
Z_{out} is the characteristic impedance of the transmission line

The reflection coefficient of an antenna is related to its impedance match with the transmission line or system it is connected to. A high reflection coefficient indicates a poor impedance match, where a significant amount of power is reflected back to the source. Conversely, a low reflection coefficient indicates a good impedance match, where most of the power is delivered to the antenna and radiated.

The value of Γ ranges from -1 to 1:
$\Gamma = 0$: No reflection; all of the wave energy is transmitted.
$\Gamma = \pm 1$: Total reflection; no wave energy is transmitted.
0<Γ<1: Partial reflection; part of the wave energy is transmitted, and part is reflected.

In the context of antennas and radio frequency systems, the reflection coefficient is typically a complex parameter. This is because it represents both the magnitude and phase shift of the reflected wave relative to the incident wave.

The complex reflection coefficient Γ can be represented as:

$$\Gamma = -|\Gamma|e^{j\theta}$$

$|\Gamma|$ is the magnitude of the reflection coefficient, representing the ratio of reflected to incident wave amplitudes.
θ is the phase angle of the reflection, showing the phase shift of the reflected wave relative to the incident wave.
j is the imaginary unit, j=$\sqrt{-1}$

3.9 Return loss

The return loss of a PCB antenna is a measure of how well the antenna is matched to the transmission line or system it is connected to. It is defined as the ratio of the power of the reflected wave to the power of the incident wave at the point of impedance mismatch, expressed in decibels (dB). The expression for return loss and S_{11} is

$$\text{Return loss} = -S_{11} = 20\log_{10}(|\Gamma|)$$

$$S_{11} = 20\log_{10}(|\Gamma|)$$

$|\Gamma|$ is the input reflection coefficient

This means that if S_{11} is a negative value (since reflection coefficient values are less than 1 in most cases), then the return loss is a positive value, indicating how much of the signal is lost due to reflection.

The return loss of a PCB antenna can be affected by various factors, such as the geometry and impedance of the antenna, the matching circuitry used to connect the antenna to the transmission line or system, and the surrounding environment.

As S_{11} and the return loss, both are related to the reflection loss, the reflection in a system can be denoted by each of these parameter. As specified above in the equations, an important point to consider is that S_{11} and return loss are oppositely related.

A high return loss or low S_{11} indicates a good impedance match between the antenna and the transmission line or system, which means that most of the power is radiated by the antenna and little is reflected back to the source. A low return loss or high S_{11} indicates an impedance mismatch, which means that some of the power is reflected back to the source and not radiated by the antenna.

3.10 VSWR

VSWR (Voltage Standing Wave Ratio) is a measure of the impedance match between an antenna and the transmission line or system it is connected to. It is defined as the ratio of the maximum voltage to the minimum voltage along a transmission line, which corresponds to the ratio of the maximum current to the minimum current. It is expressed as a unit-less value, typically between 1 and infinity.

$$\Gamma = \frac{(Z_{in} - Z_{out})}{(Z_{in} + Z_{out})}$$

$$\text{VSWR} = \frac{(1 + |\Gamma|)}{(1 - |\Gamma|)}$$

A VSWR of 1:1 represents a perfect impedance match between the antenna and transmission line, minimizing reflections and maximizing power transfer. Higher VSWR values indicate poorer matches, with a VSWR of infinity representing a complete mismatch, where all power is reflected. VSWR, return loss, and reflection coefficient are interrelated metrics used to assess impedance matching. All depend on the impedance of both the antenna and the transmission line, where mismatches can reduce antenna performance. Environmental factors, such as nearby components, ground planes, or objects, also impact the impedance match and overall efficiency of the antenna.

4 Influence of ground plane on antennas

A ground plane is a large conductive surface, typically a copper layer on the PCB's opposite side of the antenna element, providing a stable reference point and a low-impedance return path for antenna currents. It plays a crucial role in antenna characteristics by helping to reflect signals and radiate them into free space, as well as reducing the impact of nearby objects.

For proper antenna operation, the ground plane must be carefully designed, as its size, shape, and positioning significantly affect the antenna's resonant frequency, impedance, and radiation pattern. Poorly designed ground planes can introduce standing waves that interfere with signal propagation. This is especially critical for compact, PCB-based monopole antennas where the ground plane's influence is pronounced. By optimizing the ground plane, even small, integrated antennas can achieve better characteristics. Thus, a deep understanding of ground plane effects is essential for effective antenna design.

4.1 Effects of the ground plane on the antenna characteristics

The ground plane significantly impacts antenna characteristics by affecting its resonant frequency, impedance, and radiation pattern. Proper design of the ground plane is crucial, especially for compact antennas, to ensure optimal signal strength, bandwidth, and stability. In the following sections the influence of ground plane on specific antenna characteristics are demonstrated.

4.1.1 Ground plane influence on resonant frequency and bandwidth

A larger ground plane can lower the resonant frequency of an antenna. This is because a larger area increases the effective capacitance between the antenna and the ground, leading to a decrease in the resonant frequency. A larger ground plane can increase the bandwidth of the antenna. This occurs because a well-designed ground plane provides a more stable reference, which helps in achieving a more uniform current distribution along the antenna, thus allowing it to operate effectively over a broader frequency range. By providing a better reference plane, a larger ground plane improves the antenna's radiation characteristics, allowing it to radiate more efficiently across a wider frequency range.

4.1.2 Ground plane influence on radiation pattern and directivity

The ground plane significantly influences the radiation pattern of an antenna, with its effects varying based on the antenna design and application. When an antenna is positioned above a ground plane, the plane acts as a reference surface, shaping the radiation pattern. A well-designed ground plane can minimize distortion and enhance omni-directional radiation, allowing the antenna to radiate signals uniformly in all directions. Conversely, an improperly configured ground plane may introduce unwanted reflections or standing waves, negatively impacting characteristics. Ultimately, optimizing the ground plane is essential for achieving desired radiation characteristics and overall antenna efficiency.

4.1.3 Ground plane influence on gain and antenna efficiency

A larger ground plane typically enhances antenna gain by providing an expanded surface area for radiating electromagnetic waves, which improves overall radiation efficiency. When properly designed, it serves as a better reference plane, reducing ground losses and optimizing current distribution across the antenna. This leads to improved impedance matching, higher directivity, and ultimately increased gain, making the ground plane a critical element in antenna design.

4.1.4 Ground plane influence on return loss

The ground plane significantly impacts an antenna's return loss, as input impedance, VSWR, reflection coefficient, and return loss are interrelated; changes in one parameter affect the others. By providing a stable reference plane, a well-designed ground plane minimizes ground losses and reduces power reflection. Additionally, the shape and proximity of the ground plane can influence antenna characteristics: a wider ground plane may introduce directional effects that alter impedance and radiation patterns, while excessive proximity can lead to parasitic capacitance, distorting radiation patterns and diminishing efficiency.

4.2 Ground plane design

To optimize the characteristics of an antenna, it is important to carefully design the ground plane in relation to the antenna. Simulations and prototyping can help to determine the optimal ground plane dimensions and position of a given antenna. Additionally, techniques such as using matching circuits, adjusting the antenna height can also be used to improve the characteristics of the antenna.
Designing a ground plane for a PCB antenna involves several key steps. These steps are described as follows:

1. Determine the operating frequency:
   The first step in designing an RF ground plane for a PCB antenna is to determine the operating frequency. The size and shape of the ground plane depend on the frequency of operation. Once the operating frequency is determined, the size and shape of the ground plane can be calculated.

2. Calculate the ground plane size:
   The ground plane size should be at least a quarter wavelength square area at the operating frequency. For example, if the operating frequency is 2.4 GHz, the quarter wavelength is approximately 31.25 mm. Therefore, the ground plane should be at least 31.25 mm x 31.25 mm in size.

3. Choose the ground plane shape:
   The shape of the ground plane is also important. A regular shape, such as a square or rectangle, is preferred. This is because a regular shape helps to reduce edge diffraction, which can affect the radiation pattern of the antenna.

4. Determine the PCB thickness:
   In case of multilayered PCBs, the thickness of the substrate i.e., the distance between adjacent layers will also affect the antenna characteristics. Adjacent signal layer to the antenna layer shall be avoided. The adjacent layer to the antenna shall be a ground

layer and the distance to the adjacent ground layer shall be considered during the ground design.

5. Choose the ground plane material:
   Copper is the most commonly used material for ground planes due to its high conductivity. However, other conductive materials can also be used.

6. Place the ground plane beneath the antenna:
   The ground plane should be placed directly beneath the radiating element, with no traces or vias in between. This will ensure a direct and uniform ground reference for the antenna.

7. Keep clearance around the antenna:
   The clearance around the PCB antenna is also important to consider. The clearance area should be kept clear of any components or traces that could interfere with the radiation of the antenna. A clearance of at least a quarter wavelength around the antenna is recommended.

8. Connect the antenna ground plane to the host PCB ground:
   For the antenna to radiate along with the required host PCB ground, the ground plane of the antenna should be connected to the ground of the PCB. For example the top layer of the PCB which is connected to antenna should be connected to the main groundplane layer of the PCB. The different ground layers is connected using vias. The number and placement of vias depend on the frequency of operation and the thickness of the ground plane. The vias should be placed in a regular pattern and should be evenly distributed around the ground plane.

   As majorly the PCBs are made using FR4 material, the above steps does not include the substrate material. In case a different substrate other than PCB is used, the substrate material along with the ground also plays a crucial role in the antenna characteristics.

4.2.1 Conclusion

By following these steps, one can design a ground plane that provides a good ground reference for the antenna and improves its radiation pattern and impedance matching.

5 Simulation and analysis

In this section the ground plane effects on the radio module's integrated PCB antenna will be described based on simulation results. The simulation model that is described in this section is designed on base of the Proteus-III radio module with an integrated PCB antenna mounted on a two layer host PCB[1] with ground plane on both top and bottom layers.

In the figure 6, the setup of the simulation model is shown. The host PCB has a substrate thickness of 1.5 mm and two copper layers, one on each side, which act as the carrier ground plane. It is also connected to the ground plane of the radio module. The copper thickness of the carrier ground plane on the host PCB is 0.035 mm. Beneath and around the module antenna an antenna clearance area is defined where no metal shall be used, so based on the position of the module on the host PCB, the antenna clearance area is designed. The host PCB's ground plane on the radio module's antenna side has been removed to have the antenna clearance area where no copper ground or traces (metal) are used.

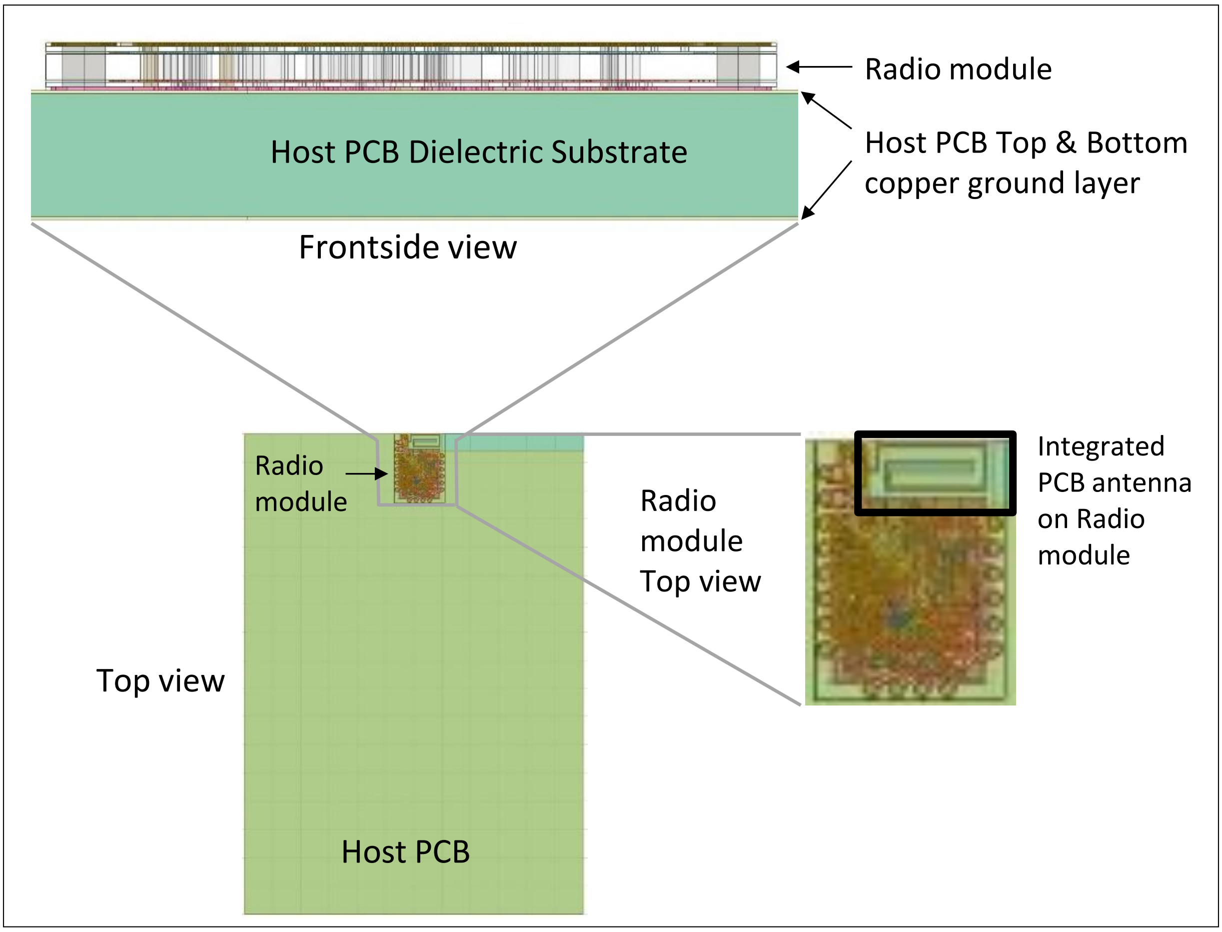

Figure 6: Simulation model

[1] The "host PCB" is the PCB where the radio module is mounted on.

Figure 7 shows the ground plane and the antenna clearance along with its measure variables. The simulation model is designed in such a way that the ground plane and antenna clearance dimensions are varied to analyze the antenna characteristics of the integrated PCB antenna of the radio module. The major antenna characteristics focused for the analysis are S_{11} parameter, peak gain and radiation pattern. The reason behind this setup is that the effect on these characteristics reflects on other antenna characteristics, as already mentioned in previous sections.

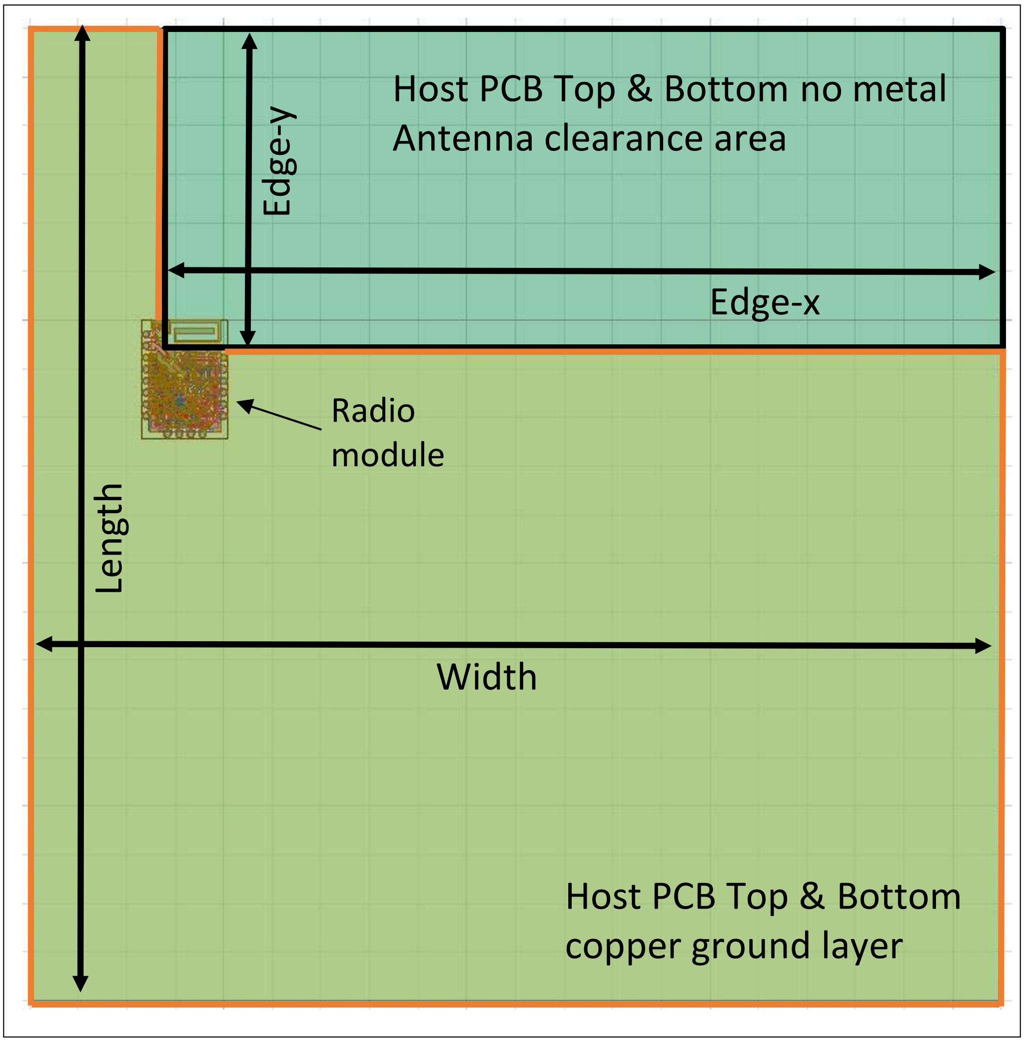

Figure 7: Ground plane dimension and clearance

5.1 Simulation setup 1

The analysis of simulation setup is divided into two parts. First part is the variation of the ground plane size with constant module position. Figure 8 shows an example for varying ground plane dimensions with minimal antenna clearance only beneath the module antenna itself and module position in the right upper corner of the host PCB. From this representation it can also be seen that the ground plane of the host PCB is much larger than the module itself. This scenario might occur in some applications or needed in some applications based on the application requirements.

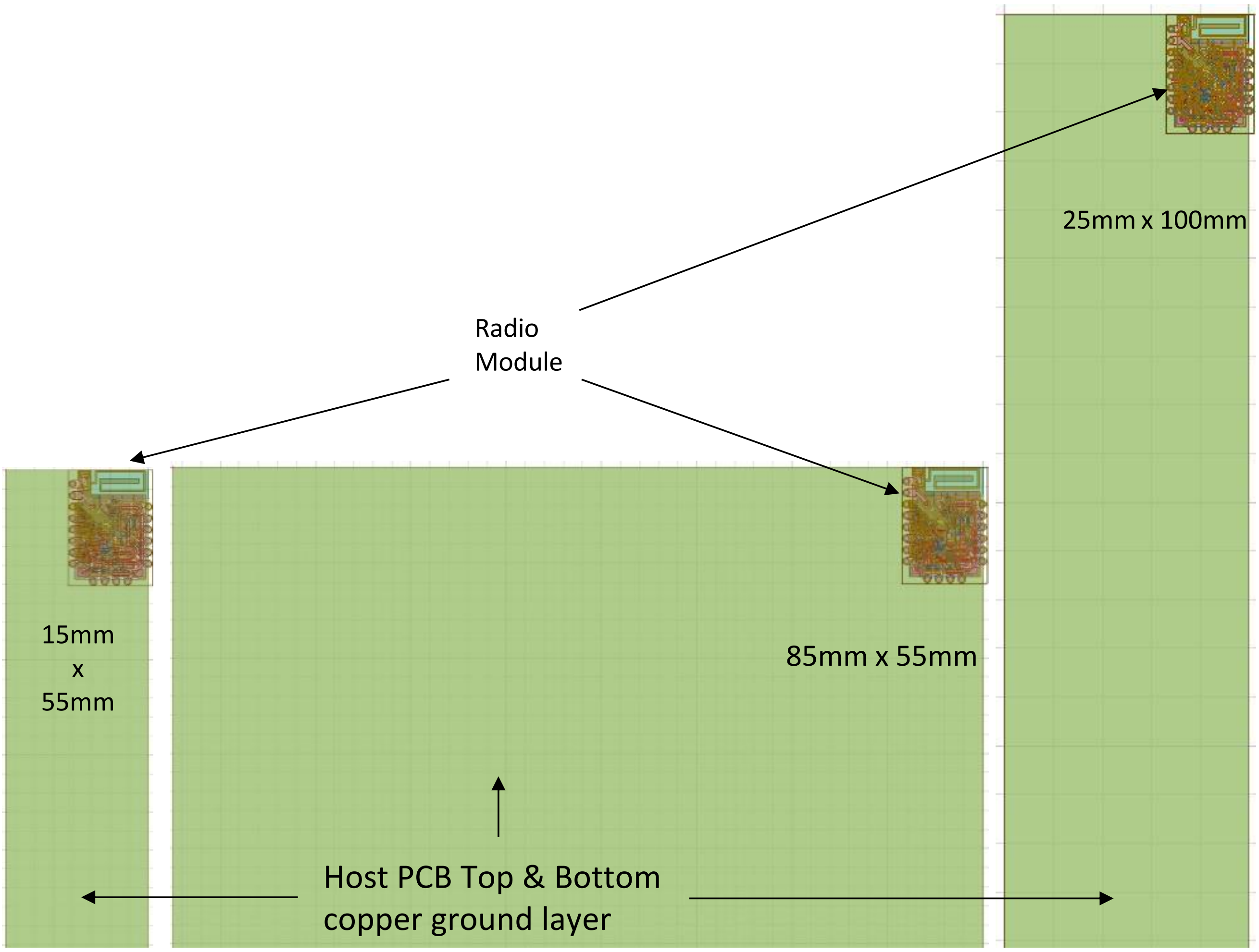

Figure 8: Varying ground plane dimension

Figure 9 shows the simulation results of the S_{11} parameter for varying ground plane dimensions with the minimal antenna clearance only beneath the module antenna itself and module is positioned in the corner of the host PCB. The S_{11} parameter is focused for 2.44 GHz as resonant frequency.

It can be observed that with increasing length of the ground plane, the S_{11} parameter decreases which is indicating a better impedance match with lower return loss. Even with very smallest width (of 15 mm) the S_{11} parameter decreases once the length of the ground plane is above 30 mm which is half wavelength of 2.44 GHz in FR4 Substrate with dielectric constant of 4.2.

Figure 9 shows further that changing the width of the ground plane with constant length also affects the S_{11} parameter of the antenna. Similar to length, around the width of 30 mm the S_{11} parameter does not fluctuate much with changes in length.

If the length and width of the ground plane is approximately the wavelength of 2.44 GHz in FR4 (around 60 mm), the S_{11} parameter decreases providing better characteristics. However the bandwidth of the antenna is not considered in this analysis which can also be affected by ground plane dimension.

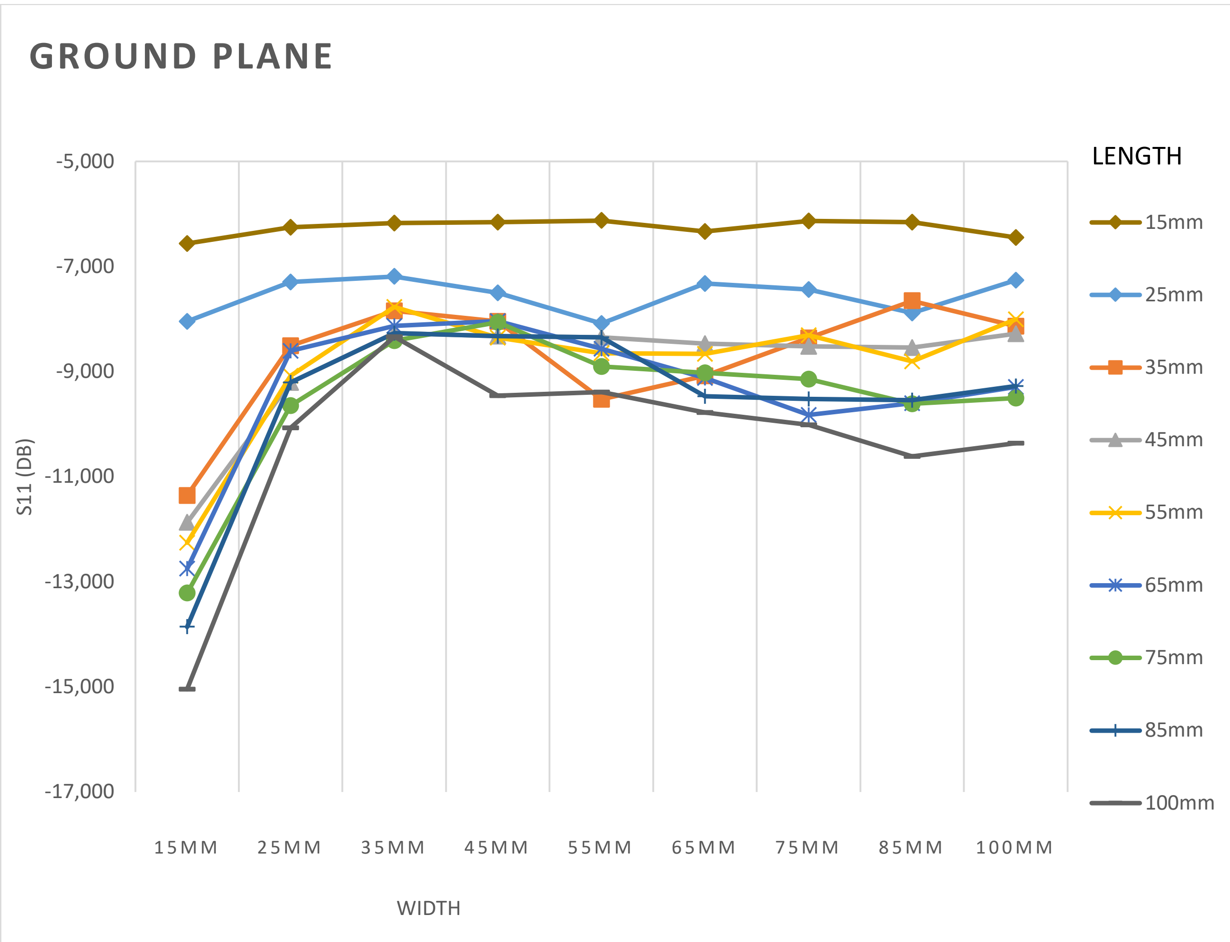

Figure 9: Ground plane dimension vs Return loss

Figure 10 shows the results of the peak gain for varying the ground plane dimensions. Please keep in mind that the peak gain does not refer to overall average gain. This means, the peak gain refers to the maximum gain of radiation pattern regardless of the directivity of the radiation pattern. Similar to the S_{11} parameter result, it is observed that with increased length of the ground plane, the peak gain increases.

Figure 10 shows that for any length of the ground plane the peak gain reaches the maximum value and minimum value based on different width of the ground plane. Any design variants with the length above 30 mm of the ground plane the peak gain value saturates around 30 mm width of the ground plane. For design variants with the length below 30 mm of the ground plane, the peak gain value tends to saturate around 60 mm width of the ground plane.

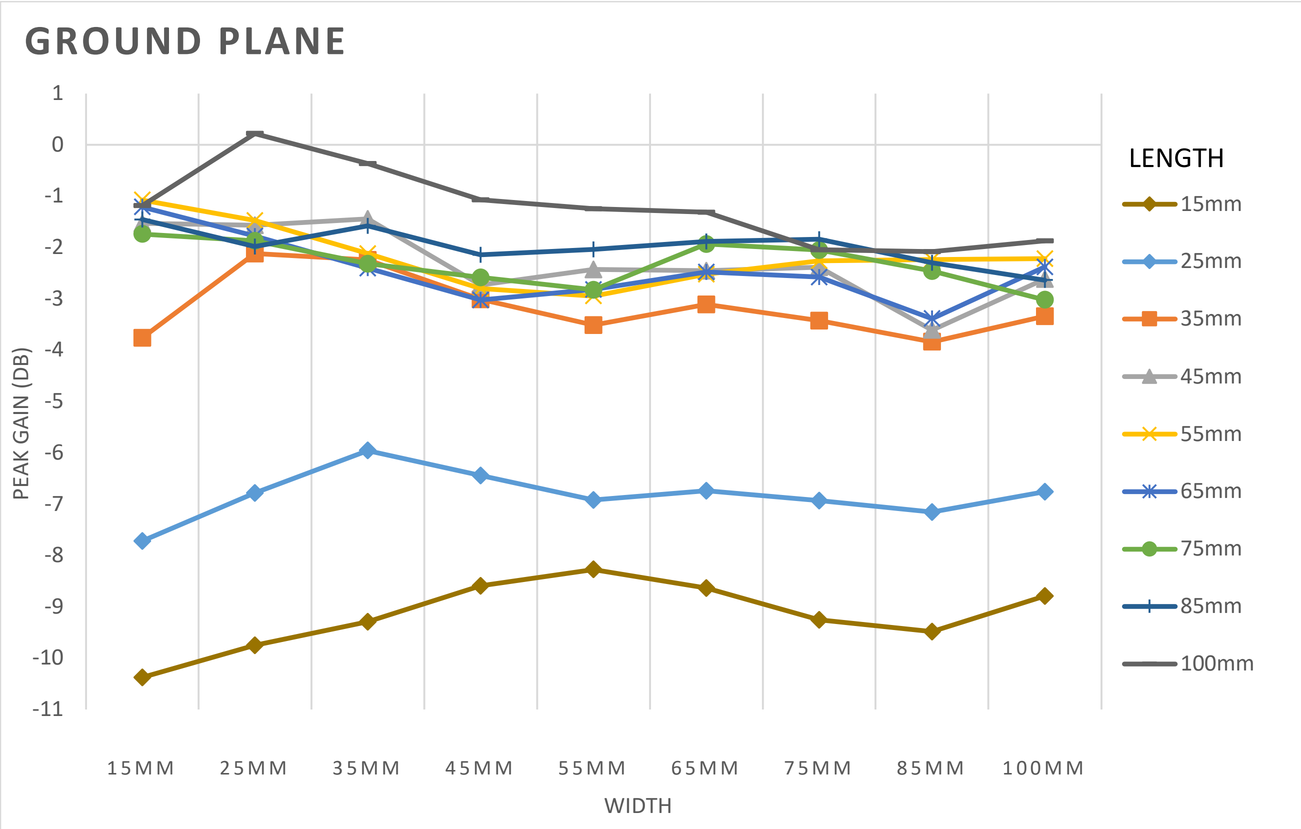

Figure 10: Ground plane dimension vs peak gain

5.2 Simulation setup 2

The second part of simulation setup is the variation of antenna clearance and module position with constant ground plane dimensions. Figure 11 shows an example of the simulation model setup. The important point to consider is that the change in the antenna clearance area and module position in turn also changes the ground plane distribution. This again influences the radio module integrated antenna characteristics which can be seen the results of the analysis.

The simulation model in figure 11 shows that increasing the antenna clearance area reduces the ground plane area as both are related to each other. In this setup the base PCB size remains same 100 mm × 100 mm. Based on the position of the module on the PCB, the antenna clearance area varies. The antenna clearance in x-direction is declared as Edge-x and antenna clearance dimension in y-direction is declared as Edge-y. Increasing the Edge-x dimension decreases the ground plane in x-direction. Increasing the Edge-y dimension decreases the ground plan in y-direction.

In addition to that the proximity of the ground plane area which acts as the reference reflector changes with the change in antenna clearance area. Due to this the significant change in the radiation pattern and gain can be observed.

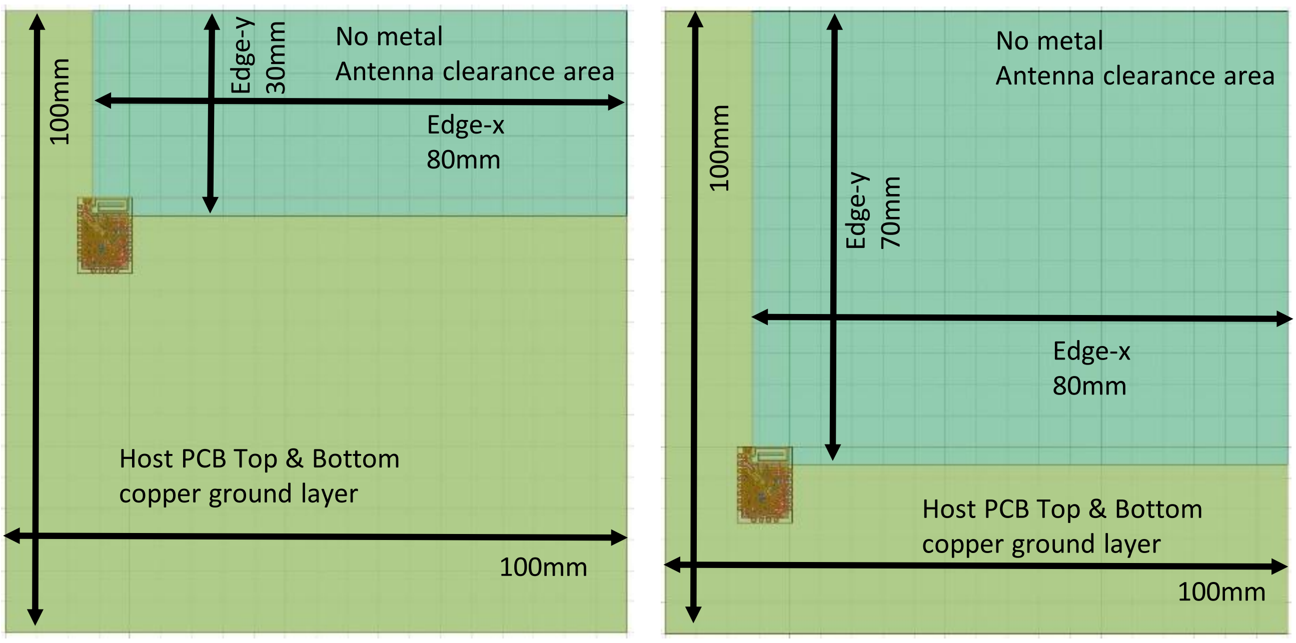

Figure 11: Varying antenna clearance and module position

Figure 12 shows the results of the S_{11} parameter for varying the antenna clearance dimensions along with changing module positions with constant PCB dimension of 100 mm × 100 mm. As the groundplane size and antenna clearance area is related to eachother, the ground plane area varies according to the antenna clearance dimensions Edge-y and Edge-y. With the Edge-x and Edge-y dimension of 0mm the module protudes out of the base PCB and it does not comply with the design guideline of the module for antenna clearance. So the Edge-x and Edge-y minimum values is taken as 10mm.

In figure 12, it can be observed that increasing the Edge-y value shows a trend of decreasing in the S_{11} parameter value. When the Edge-x value reaches around 10mm, 40mm and 65mm, the S11 parameters values decrease indicating better return loss. These values on the otherhand relates to the ground plane dimension of approximately 30 mm and 60 mm in x-direction.

The minimum value of the S_{11} parameter is achieved at Edge-x, Edge-y values around 70 mm and 40mm. These values refers to the ground plane dimension of 30 mm, 60 mm, 90 mm. As it can be seen the values mentioned are roughly the factor of quarter, half and full wavelength for 2.44 GHz in FR4 substrate. Due to that, designing a ground plane with dimensions in the factors of resonant frequency wavelength is recommended.

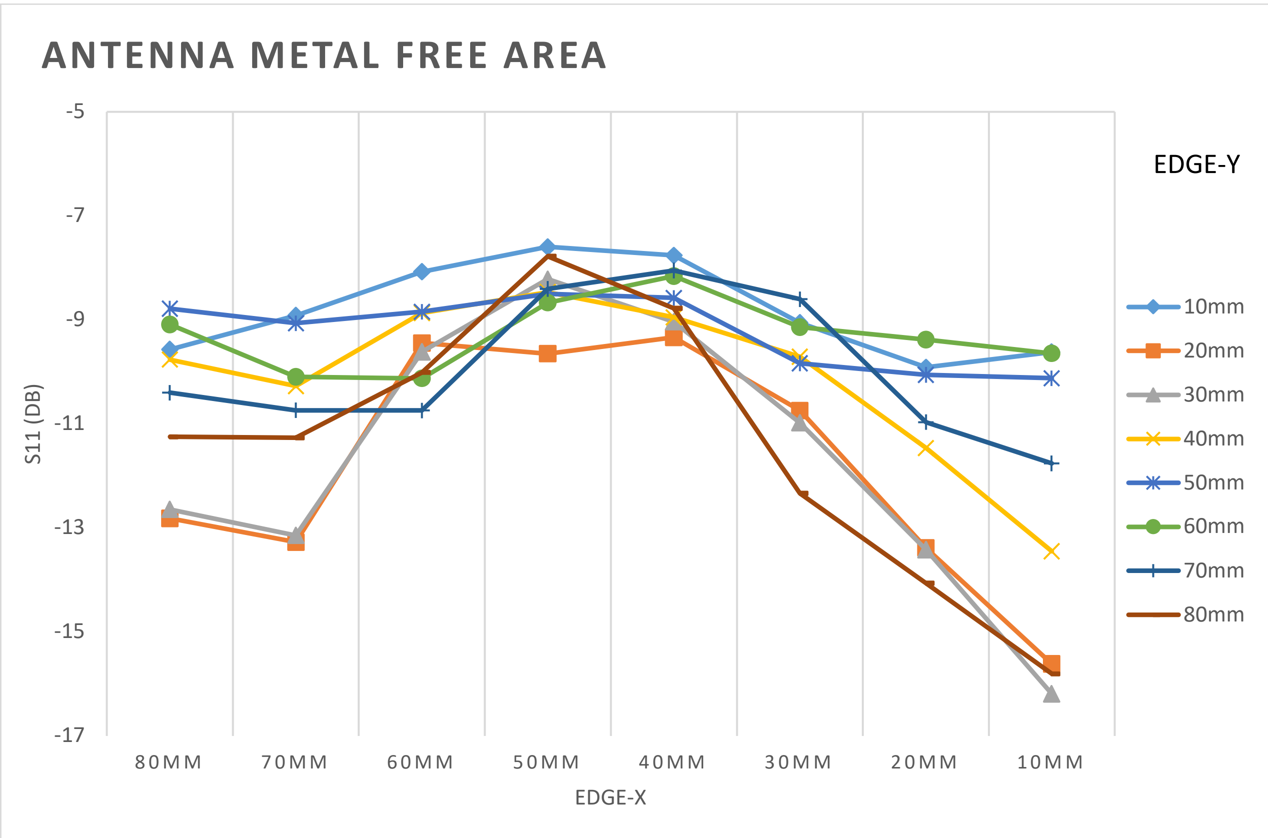

Figure 12: Antenna clearance dimension vs return loss

Figure 13 shows the results of the peak gain for varying antenna clearance dimensions along with changing module positions with constant ground plane dimension of 100 mm × 100 mm. From the observation of the result in figure 13 a very linear trend can be observed. Increasing the Edge-x and Edge-x values result in increasing the peak gain value. Although increasing the Edge-x and Edge-y values reduce the ground plane area, significant increase in the peak gain is observed from the results.

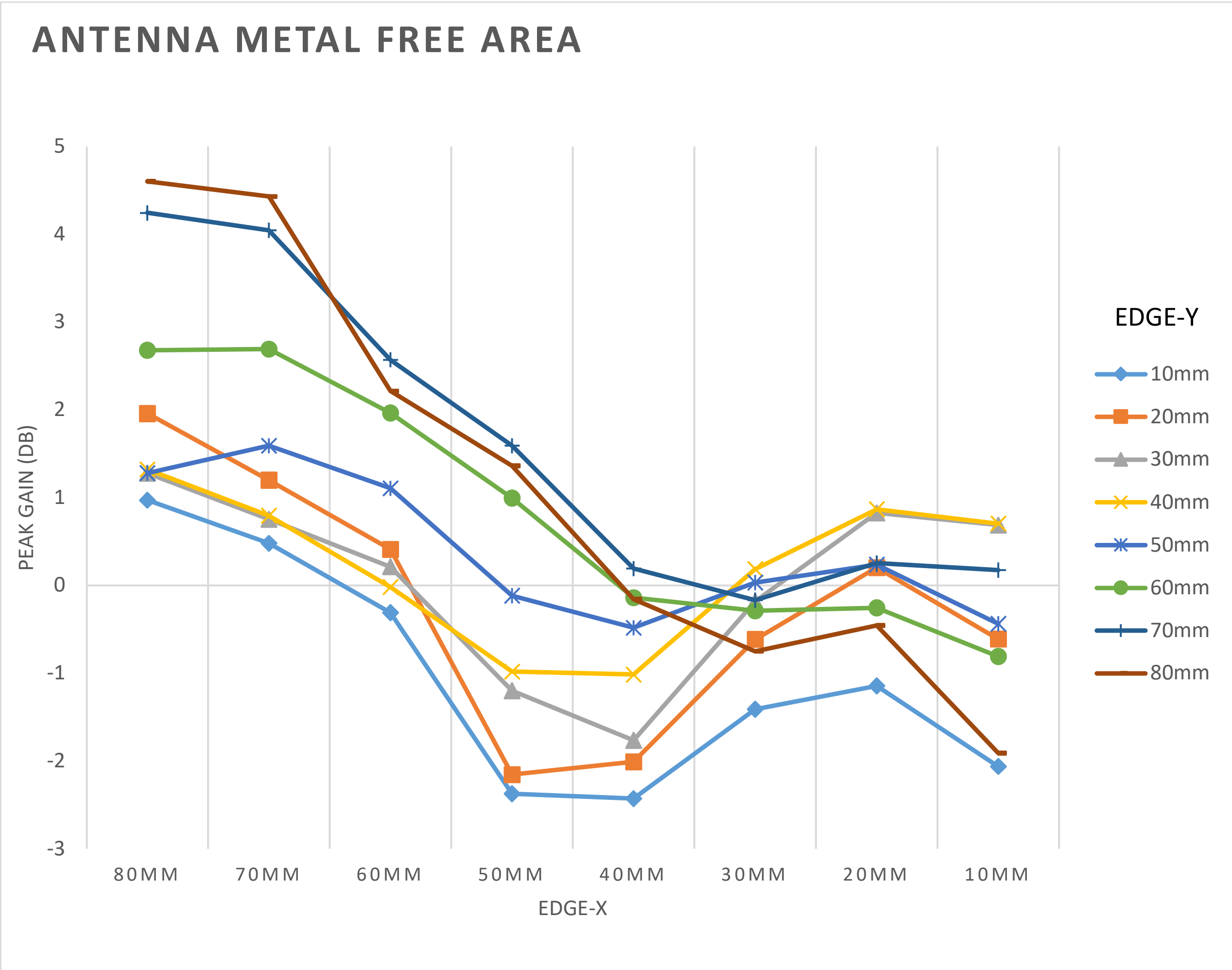

Figure 13: Antenna clearance dimension vs peak gain

A very important factor here to be noted is that the peak gain achieved in these variations comes with a very unidirectional radiation pattern. This is due to the large antenna clearance area with very close proximity of ground plane adjacent to the antenna. This effect leads the antenna to radiate in unidirectional manner focusing the radiation towards the antenna clearance area. Adjacent ground plane to the antenna significant reduces the gain on the those directions and increases the peak gain in one direction. This phenomenon is similar to the additional reflector elements in the Yagi antenna, whereas in the simulation model the adjacent ground plane acts as the reflectors. So the antenna clearance area has significant effect on the peak gain through the radiation pattern of the antenna.

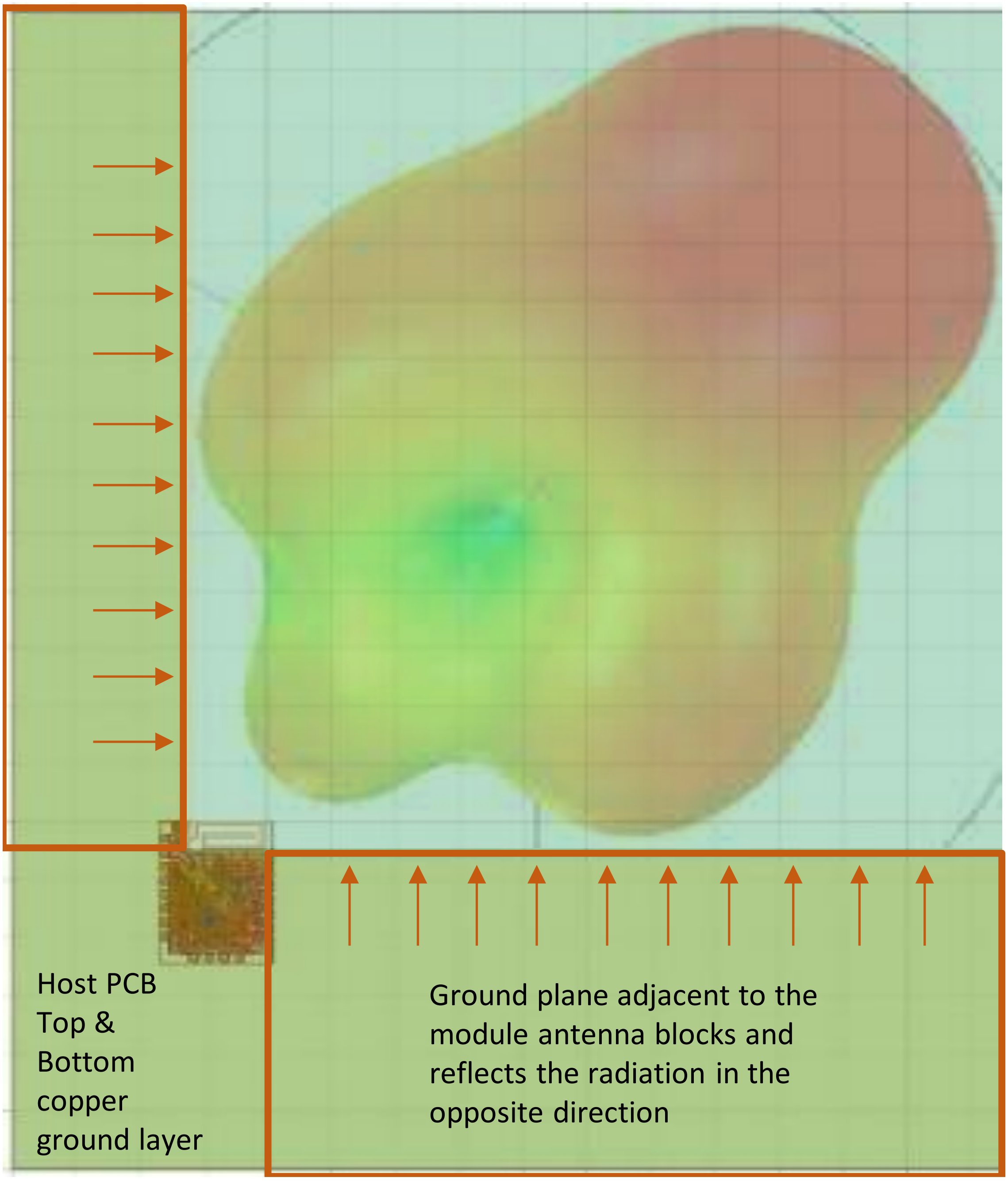

Figure 14: Undirectional radiation pattern

From the analysis a few examples are used to illustrate the radiation pattern. In the figure 15 the radiation pattern with the same antenna clearance area with different ground plane in simulation are shown. The design A (15 mm × 55 mm) is observed to have almost an omni-directional gain in xz-plane. By increasing the width from 15 mm to 85 mm, the design is changed from A to B. In the design B, the ground plane is extended more in the opposite direction (-x,-y) to the direction of the antenna (x,y). The radiation pattern also results in the similar manner with a higher radiation in the direction (-x-y).

By changing the design variant A to C, the length is increased from 55 mm to 100 mm and width is increased from 15 mm to 25 mm. In this design the radiation pattern is splitted to have two lobes. The radiation lobe in the direction (x,-y) of the extended ground plane exhibit higher radiation than the antenna direction (x,y). This is due to ground plane outweighing the antenna. This is because the ground plane acts as the second radiator element for the quarter wave antenna.

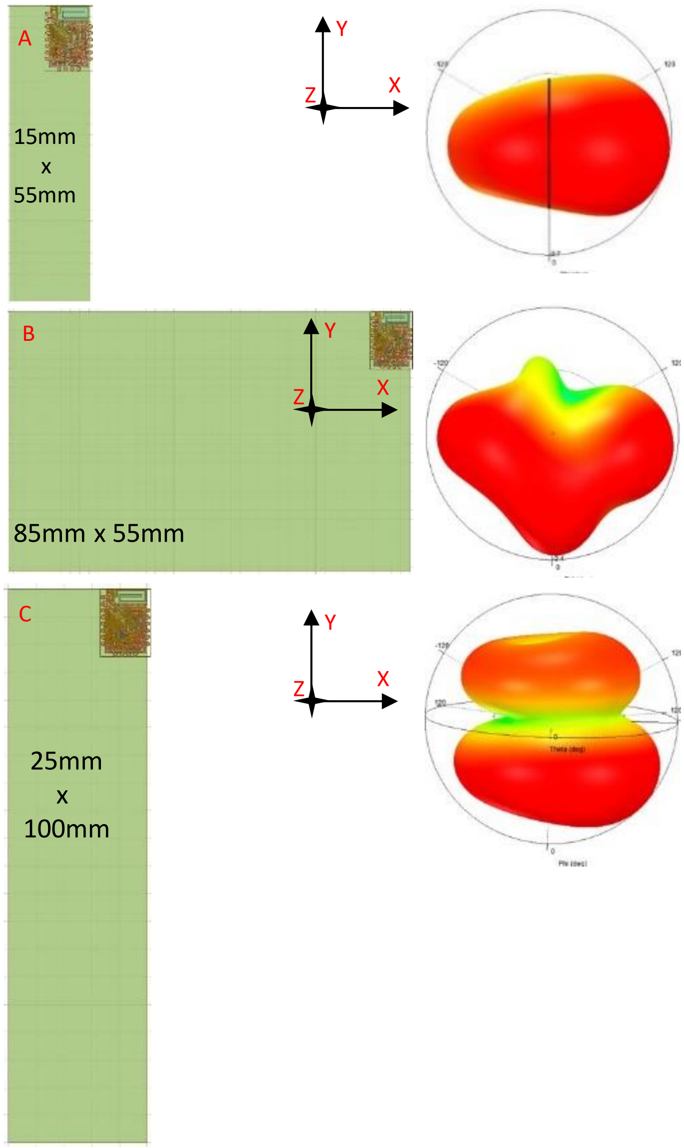

Figure 15: Radiation pattern - simulation setup 1

Figure 16 shows simulation results with different antenna clearance areas while keeping the same PCB dimension.. In both design variants D and E, the PCB dimension remains the same (100 mm × 100 mm). The antenna clearance area is modified in y-direction and the ground plane area changes accordingly. The edge-y is changed from 30 mm to 70 mm. In both variation D and E the radiation pattern is unidirectional. This is due to the presence of the adjacent ground planes on both side of the antenna. It constricts the effective radiation on these directions (-x and -y direction).

The antenna clearance area on one corner serves as the pathway for radiation. The design variant E with larger metal free area (80 mm × 70 mm) has a more narrow radiation pattern with higher peak gain than design variant D. The radiated power can be focused more in one direction, and the peak gain of the variant E is higher w.r.t. variant D. But in terms of the radiation pattern, the variant D has better directivity in other directions. So if the end application needs a more directed radiation pattern the design similar to variant E shall be preferred.

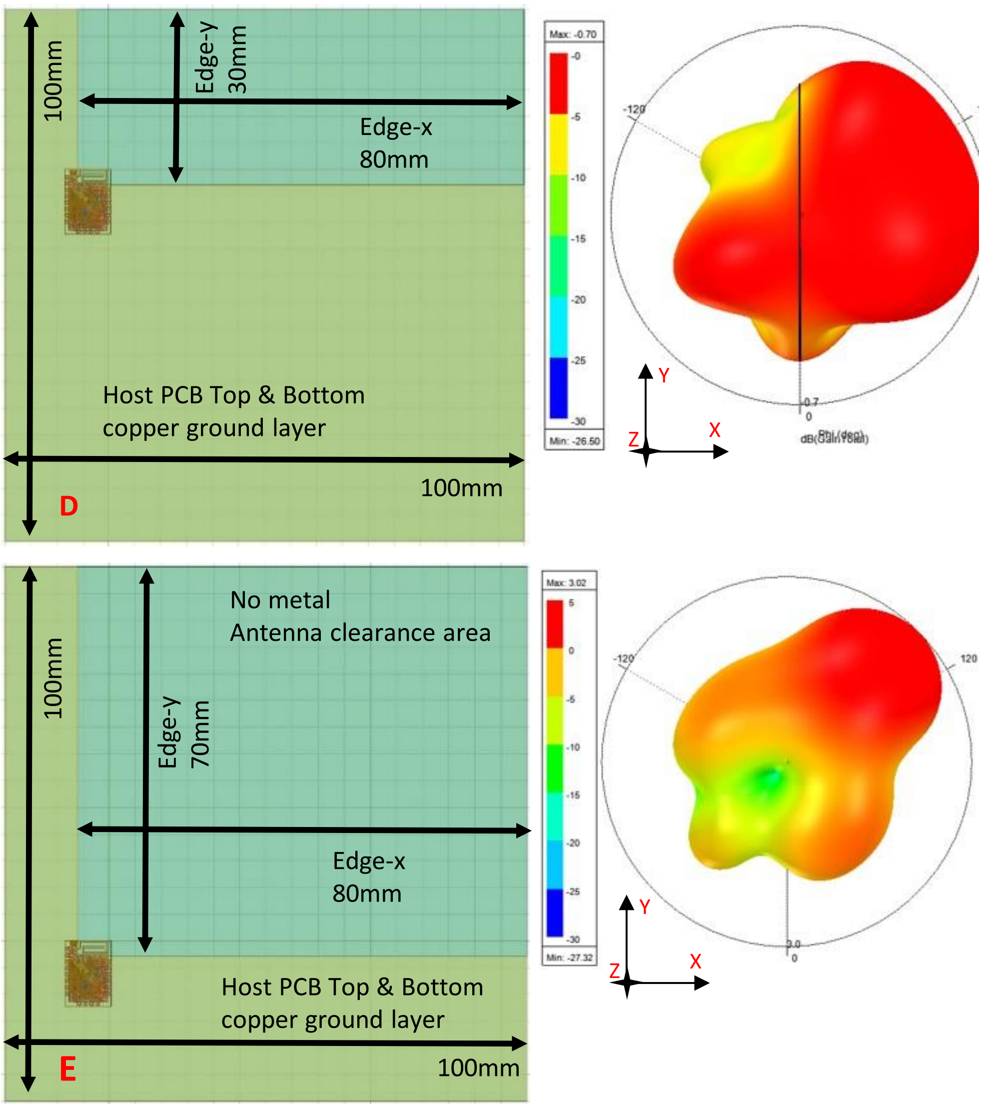

Figure 16: Radiation pattern - simulation setup 2

6 Testing and verification

This section illustrates the radiation pattern of the selected designs from the previous section. The design variant A to E from the previous sections are realized on the PCBs. The radio module Proteus-III is used for this realization. The boards are tested using the shielded OTA chamber to compare the measured radiation results with the simulated results.
In the figure 17 the radiation patterns with same antenna clearance area with different ground planes on the PCB are shown. As the credibility of radiation pattern from a simulation can be questionable, the testing and realization of the simulation on the PCB were necessary. This helps the engineers to rely on the integrity of the results from the simulation. As shown in the figure 17 and 18, the radiation pattern test from OTA chamber provide very similar radiation results compared to the simulation.

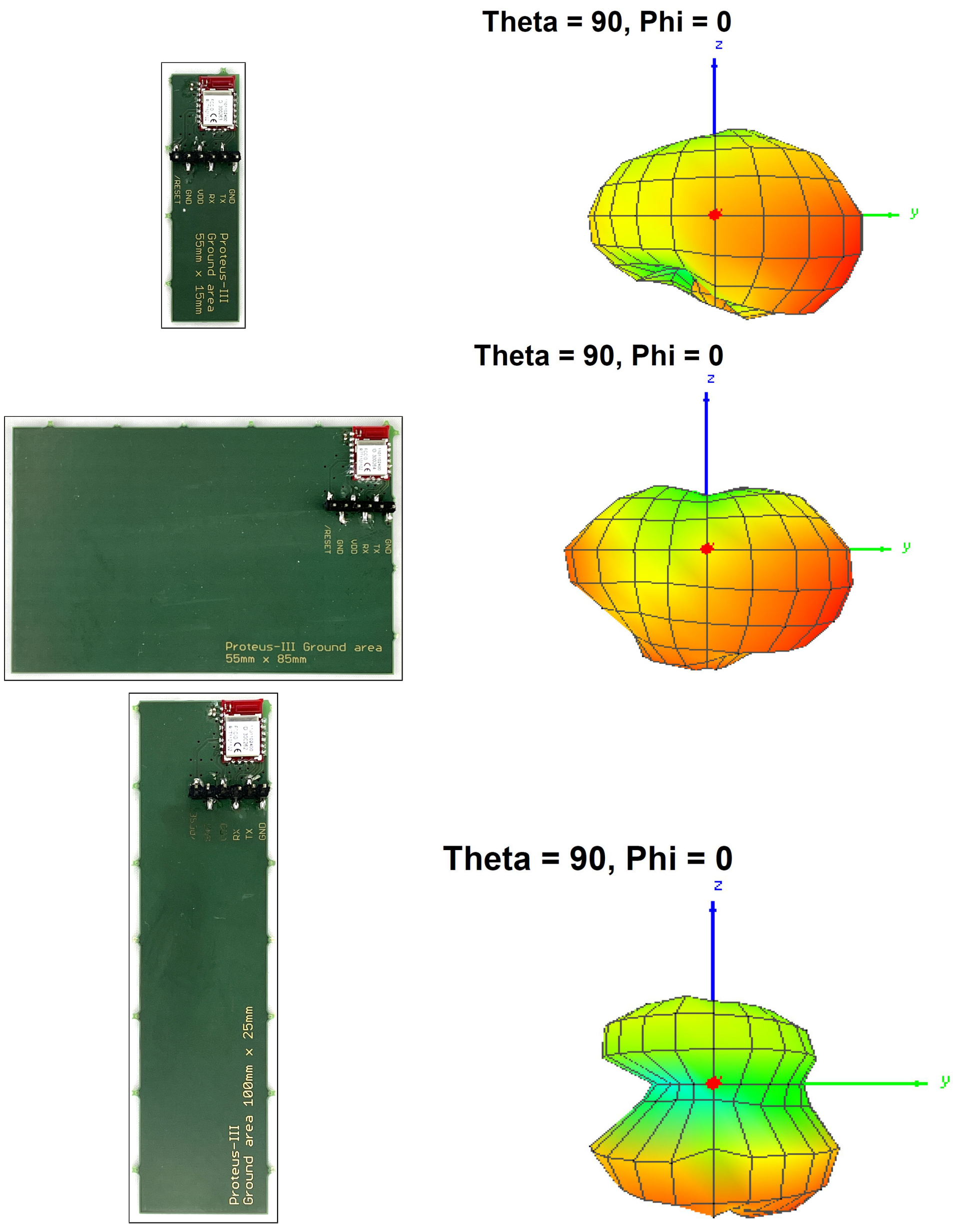

Figure 17: Measurement - simulation setup 1

In the figure 15 the radiation pattern with same ground plane area with different antenna clearance on the PCB are shown.

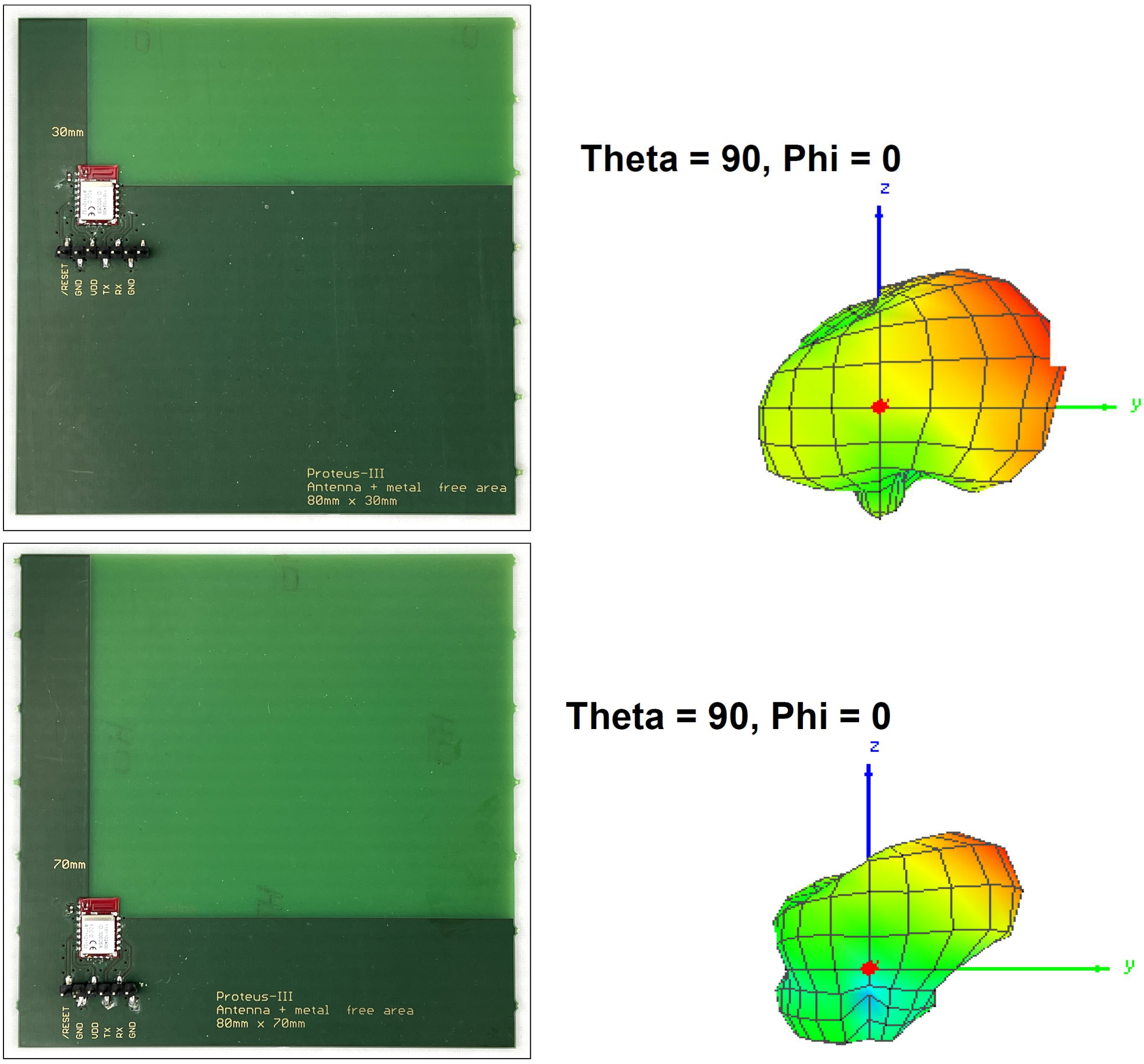

Figure 18: Measurement - simulation setup 2

6.1 Simulation vs Measurement

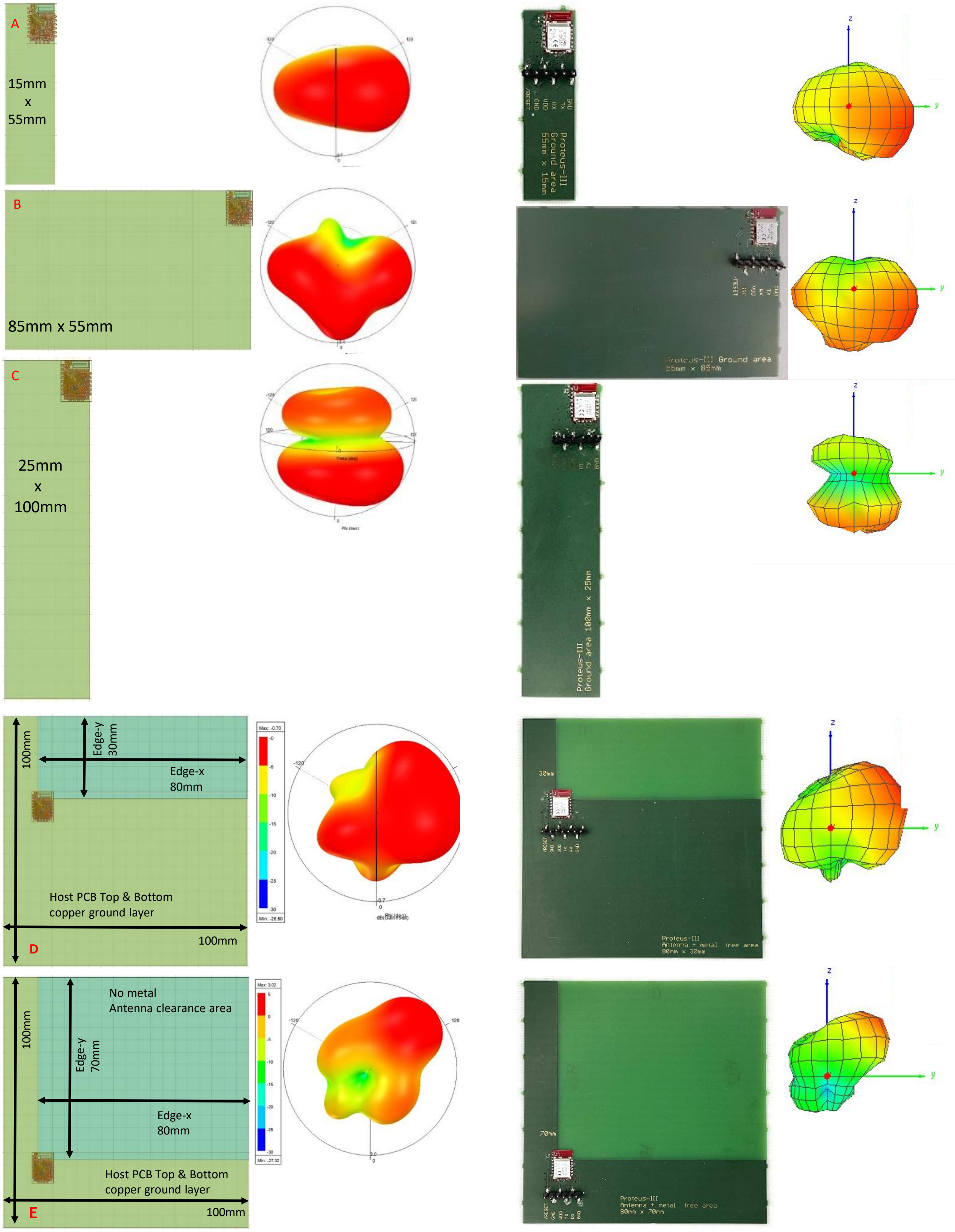

Figure 19: Comparision simulation and measurement

7 Summary

This application note provides information regarding the effects of the ground plane on the PCB antenna and guidelines for design optimization of the ground plane for desired PCB antenna characteristics. As mentioned in this application note there are numerous antennas which rely majorly on the ground plane for effective radiation, especially if they are small in size. This is due to the miniaturized and compact antenna design where the ground plane acts as an additional radiator and reflector element supporting the antenna. This is the reason the ground plane can significantly affect the characteristics of a PCB antenna.

In addition to the dimensions of the ground plane, the proximity of the ground plane can cause changes in the radiation pattern, impedance, resonant frequency, and efficiency of the antenna. To optimize the characteristics of a PCB antenna, it is essential to consider the effects of the ground plane. It is also important to implement techniques to design proper ground plane dimension, antenna clearance, keeping the ground plane away from the antenna and using a proper matching network for impedance matching. By understanding the impacts of the ground plane on the PCB antenna and implementing appropriate design techniques, engineers can fine tune the antenna's characteristics which in return supports the overall performance of the electronic device.

8 Important notes

The Application Note and its containing information ("Information") is based on Würth Elektronik eiSos GmbH & Co. KG and its subsidiaries and affiliates ("WE eiSos") knowledge and experience of typical requirements concerning these areas. It serves as general guidance and shall not be construed as a commitment for the suitability for customer applications by WE eiSos. While WE eiSos has used reasonable efforts to ensure the accuracy of the Information, WE eiSos does not guarantee that the Information is error-free, nor makes any other representation, warranty or guarantee that the Information is completely accurate or up-to-date. The Information is subject to change without notice. To the extent permitted by law, the Information shall not be reproduced or copied without WE eiSos' prior written permission. In any case, the Information, in full or in parts, may not be altered, falsified or distorted nor be used for any unauthorized purpose.

WE eiSos is not liable for application assistance of any kind. Customer may use WE eiSos' assistance and product recommendations for customer's applications and design. No oral or written Information given by WE eiSos or its distributors, agents or employees will operate to create any warranty or guarantee or vary any official documentation of the product e.g. data sheets and user manuals towards customer and customer shall not rely on any provided Information. THE INFORMATION IS PROVIDED "AS IS". CUSTOMER ACKNOWLEDGES THAT WE EISOS MAKES NO REPRESENTATIONS AND WARRANTIES OF ANY KIND RELATED TO, BUT NOT LIMITED TO THE NON-INFRINGEMENT OF THIRD PARTIES' INTELLECTUAL PROPERTY RIGHTS OR THE MERCHANTABILITY OR FITNESS FOR A PURPOSE OR USAGE. WE EISOS DOES NOT WARRANT OR REPRESENT THAT ANY LICENSE, EITHER EXPRESS OR IMPLIED, IS GRANTED UNDER ANY PATENT RIGHT, COPYRIGHT, MASK WORK RIGHT, OR OTHER INTELLECTUAL PROPERTY RIGHT RELATING TO ANY COMBINATION, MACHINE, OR PROCESS IN WHICH WE EISOS INFORMATION IS USED. INFORMATION PUBLISHED BY WE EISOS REGARDING THIRD-PARTY PRODUCTS OR SERVICES DOES NOT CONSTITUTE A LICENSE FROM WE eiSos TO USE SUCH PRODUCTS OR SERVICES OR A WARRANTY OR ENDORSEMENT THEREOF.

The responsibility for the applicability and use of WE eiSos' components in a particular customer design is always solely within the authority of the customer. Due to this fact it is up to the customer to evaluate and investigate, where appropriate, and decide whether the device with the specific characteristics described in the specification is valid and suitable for the respective customer application or not. The technical specifications are stated in the current data sheet and user manual of the component. Therefore the customers shall use the data sheets and user manuals and are cautioned to verify that they are current. The data sheets and user manuals can be downloaded at *www.we-online.com*. Customers shall strictly observe any product-specific notes, cautions and warnings. WE eiSos reserves the right to make corrections, modifications, enhancements, improvements, and other changes to its products and services at any time without notice.

WE eiSos will in no case be liable for customer's use, or the results of the use, of the components or any accompanying written materials. IT IS CUSTOMER'S RESPONSIBILITY TO VERIFY THE RESULTS OF THE USE OF THIS INFORMATION IN IT'S OWN PARTICULAR ENGINEERING AND PRODUCT ENVIRONMENT AND CUSTOMER ASSUMES THE ENTIRE RISK OF DOING SO OR FAILING TO DO SO. IN NO CASE WILL WE EISOS BE LIABLE FOR CUSTOMER'S USE, OR THE RESULTS OF IT'S USE OF THE COMPONENTS OR ANY ACCOMPANYING WRITTEN MATERIAL IF CUSTOMER TRANSLATES, ALTERS, ARRANGES, TRANSFORMS, OR OTHERWISE MODIFIES THE INFORMATION IN ANY WAY, SHAPE OR FORM.

If customer determines that the components are valid and suitable for a particular design and wants to order the corresponding components, customer acknowledges to minimize the risk of loss and harm to individuals and bears the risk for failure leading to personal injury or death due to customers usage of the components. The components have been designed and developed for usage in general electronic equipment only. The components are not authorized for use in equipment where a higher safety standard and reliability standard is especially required or where a failure of the components is reasonably expected to cause severe personal injury or death, unless WE eiSos and customer have executed an agreement specifically governing such use. Moreover WE eiSos components are neither designed nor intended for use in areas such as military, aerospace, aviation, nuclear control, submarine, transportation, transportation signal, disaster prevention, medical, public information network etc. WE eiSos must be informed about the intent of such usage before the design-in stage. In addition, sufficient reliability evaluation checks for safety must be performed on every component which is used in electrical circuits that require high safety and reliability functions or performance. COSTUMER SHALL INDEMNIFY WE EISOS AGAINST ANY DAMAGES ARISING OUT OF THE USE OF THE COMPONENTS IN SUCH SAFETY-CRITICAL APPLICATIONS.

List of Figures

List of Tables

Contact
Würth Elektronik eiSos GmbH & Co. KG
Division Wireless Connectivity & Sensors

Max-Eyth-Straße 1
74638 Waldenburg
Germany

Tel.: +49 651 99355-0
Fax.: +49 651 99355-69
www.we-online.com/wireless-connectivity