



Guidelines for PCB design on STM32WBA5MMG

Introduction

The [STM32WBA5MMG](#) is a module for Bluetooth® LE and IEEE 802.15.4 low-power wireless applications. Crystals and passive components for antenna matching and power supply filtering are integrated inside the device. The module has an internal antenna, but it is also possible to connect an external antenna on or off the board. This document aims to give recommendations for the schematics and layout of applications that use this device. For more details, refer to the connectivity expansion board [B-WBA5M-WPAN](#).

1 General information

The [STM32WBA5MMG](#) module is based on the RF microcontroller [STM32WBA55UG](#). For a complete description of its features and characteristics, refer to the datasheets [1] of the SoC and [2] of the module.

For information on Arm® Cortex® cores, refer to the relevant Cortex® technical reference manual, available from the www.arm.com website.

For information on Bluetooth® refer to the www.bluetooth.com website.



Note: Arm and Cortex are registered trademarks of Arm Limited (or its subsidiaries or affiliates) in the US and/or elsewhere.

The Arm word and logo are trademarks of Arm Limited (or its subsidiaries) in the US and/or elsewhere. All rights reserved.

Reference documents

- [1] Datasheet *Multiprotocol wireless 32-bit MCU Arm®-based Cortex®-M33 with TrustZone®, FPU, Bluetooth® 5.4 and IEEE 802.15.4 radio solution* (DS14127)
- [2] Datasheet *Bluetooth® LE and IEEE 802.15.4 radio module* (DS14801)
- [3] Application note *RF matching network design guide for STM32WL Series* (AN5457)
- [4] [STM32WBA5M connectivity expansion board](#) on st.com webpage
- [5] Application note *How to develop RF hardware using STM32WBA5xxx MCUs* (AN5948)

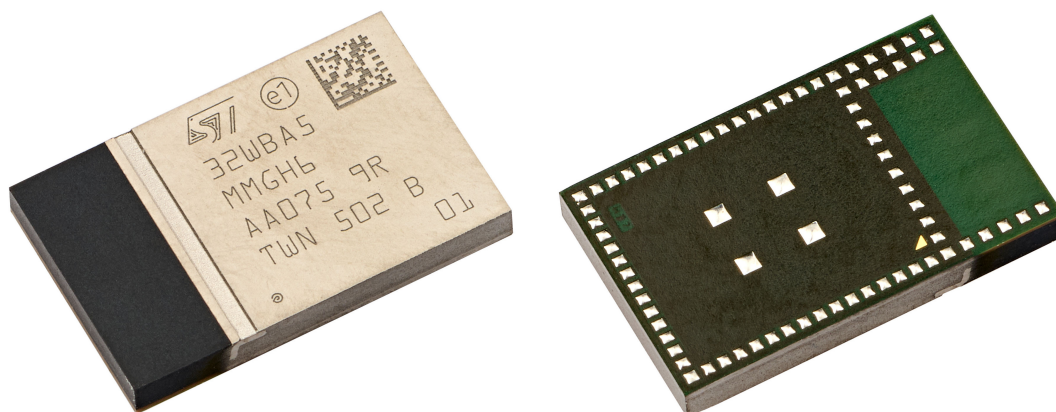
Acronyms

RF	Radio frequency
SMA	Coaxial RF connector (subminiature version A)
SoC	System-on-chip

2 General considerations

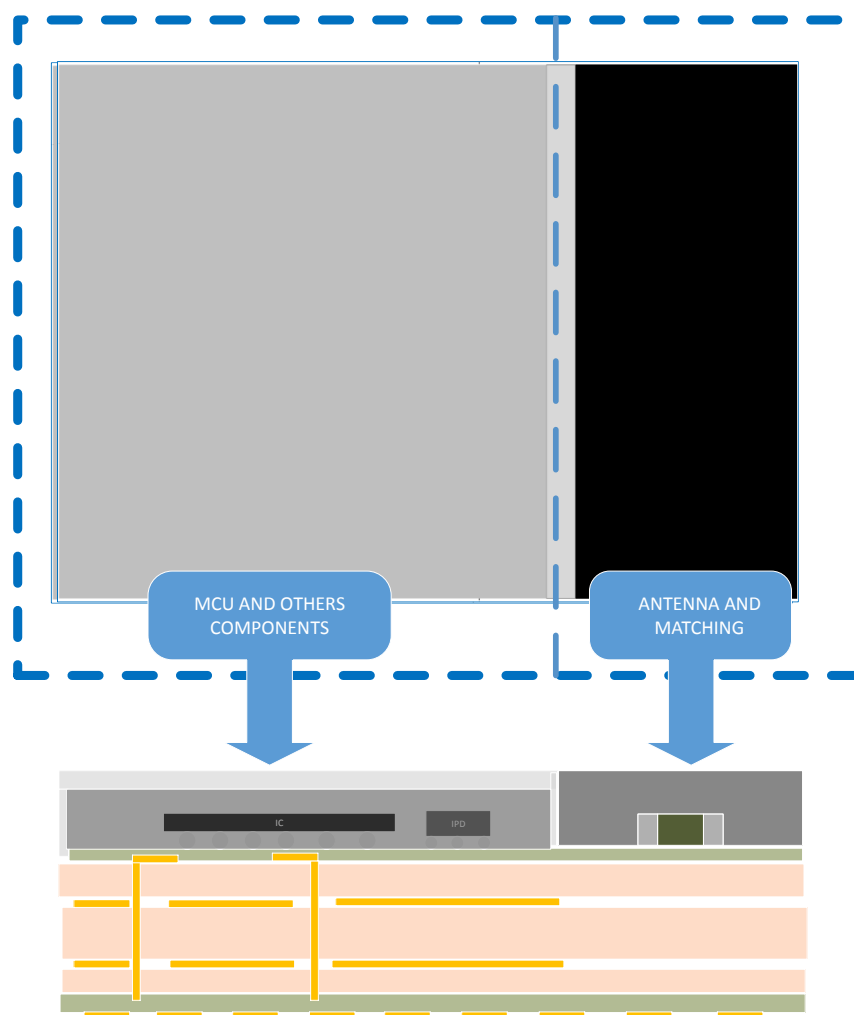
The Figure 2 and Figure 3 show that the module is divided in two parts. The SoC and other components are located under the clear shielding part of the cover. The antenna is covered by the black non-shielding area.

Figure 1. Module top and bottom view



DT79021V1

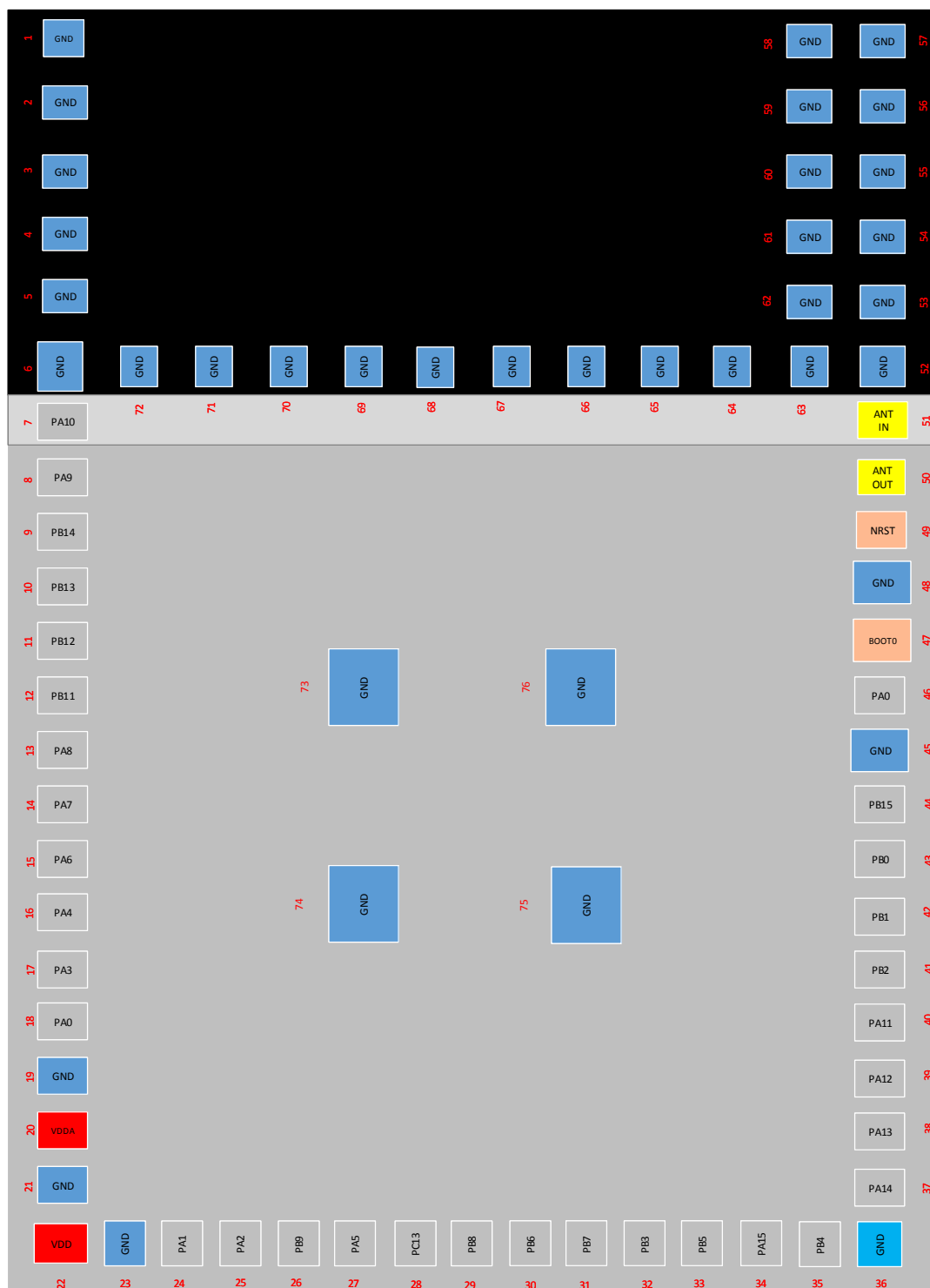
Figure 2. Module general view



DT79022V1

In the figure below, the module pinout is shown:

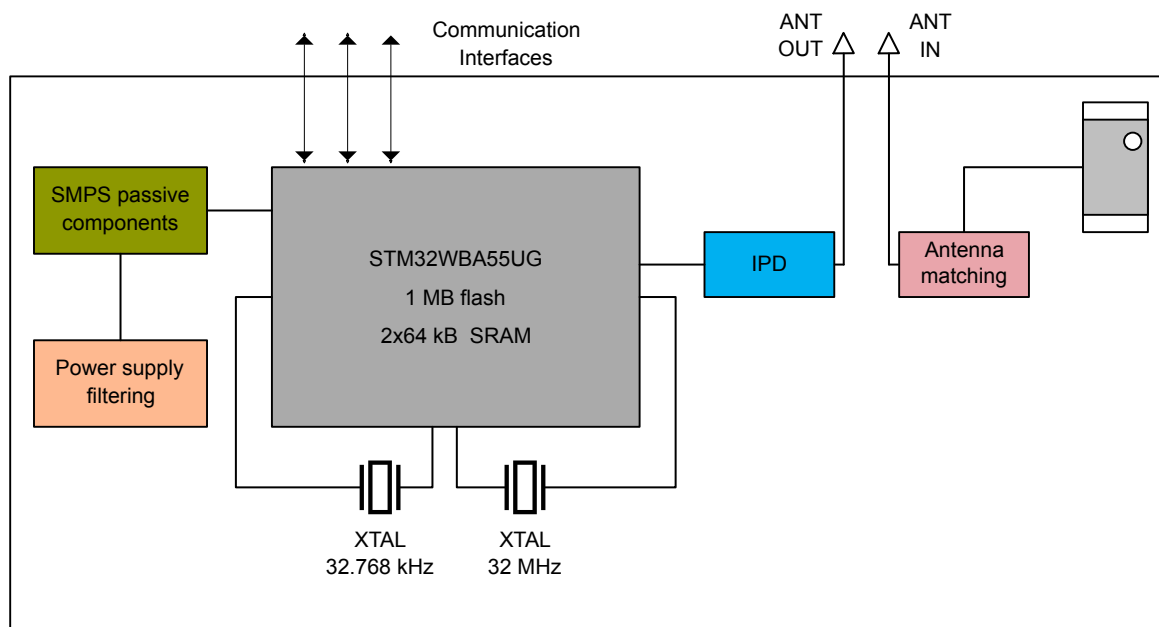
Figure 3. Top view showing the pins below



DT79023V1

The module block diagram is shown below:

Figure 4. Module block diagram



DTT9024V1

3 Basics of antenna parameters

3.1 dBm and Watt relation

The power of system transmission is expressed in milliwatts (mW). Generally it is preferred to use a log scale to express very large or very small numbers in a simple form. The dBm relates to 1 mW and Figure 5 gives some basic translations.

The formula to calculate P in mW from x dBm is:

$$P = 1mW \times 10^{x/10}$$

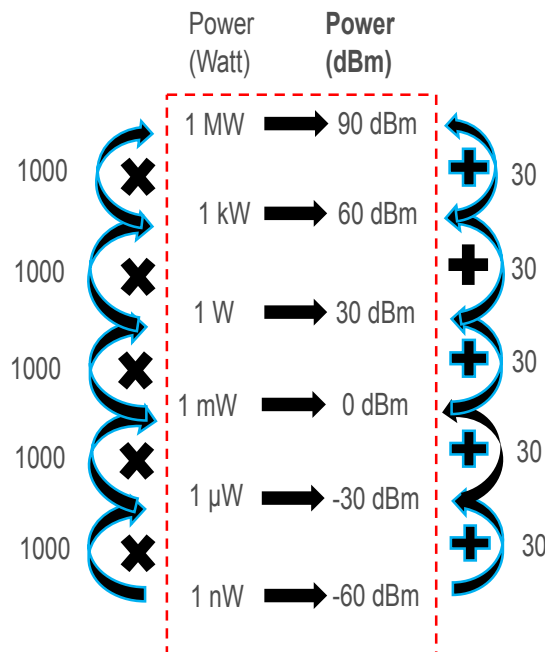
In the same way, find the power x in dBm from P in mW:

$$x = 10 \times \log(P/1mW)$$

The dBm formula and dBmW relation are as in the following:

$$P_{dBmW} = 10 \times \log\left(\frac{P_W}{1mW}\right) = 10 \times \log(P_W) + 30$$

Figure 5. Some basic correspondences between watt and dBm



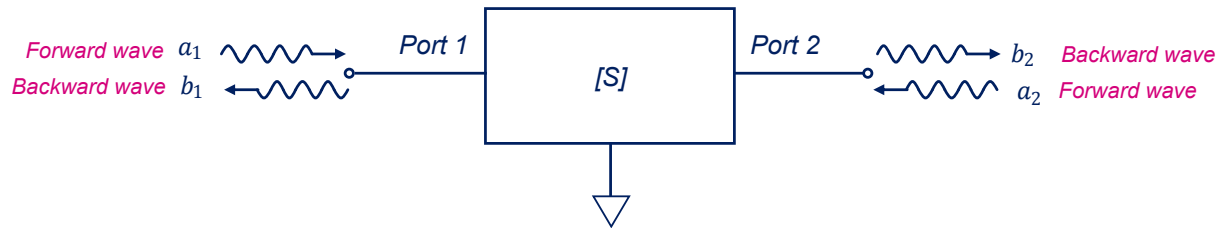
3.2

S11 parameters

S11 parameters are a mathematical representation used to describe the relationship between incident and reflected waves in a theoretical N-port system. Consider a simplified scenario involving a system with a single input port, where a signal is transmitted, and its reflection can be observed. A common example of such a one-port system is an antenna.

S parameters give an indication of the power transferred between ports or terminals in a communication system. S_{YX} gives the power that is received by Y after transmission from X. If X transmits 0 dB and Y receives -10 dB then $S_{YX} = -10$ dB. The negative value indicates a loss. Systems can be compared by looking at the S value. The closer the (negative) value is to 0, the lower the loss.

Figure 6. S-parameters view black box



$$S_{11} = \frac{\text{Reflected}}{\text{Incident}} = \frac{b_1}{a_1} \Big|_{a_2=0} \quad S_{12} = \frac{\text{Transmitted}}{\text{Incident}} = \frac{b_1}{a_2} \Big|_{a_1=0}$$

$$S_{21} = \frac{\text{Transmitted}}{\text{Incident}} = \frac{b_2}{a_1} \Big|_{a_2=0} \quad S_{22} = \frac{\text{Reflected}}{\text{Incident}} = \frac{b_2}{a_2} \Big|_{a_1=0}$$

DT79028V1

S_{11} is the reflection coefficient or return loss. It tells us how much power is effectively radiated.

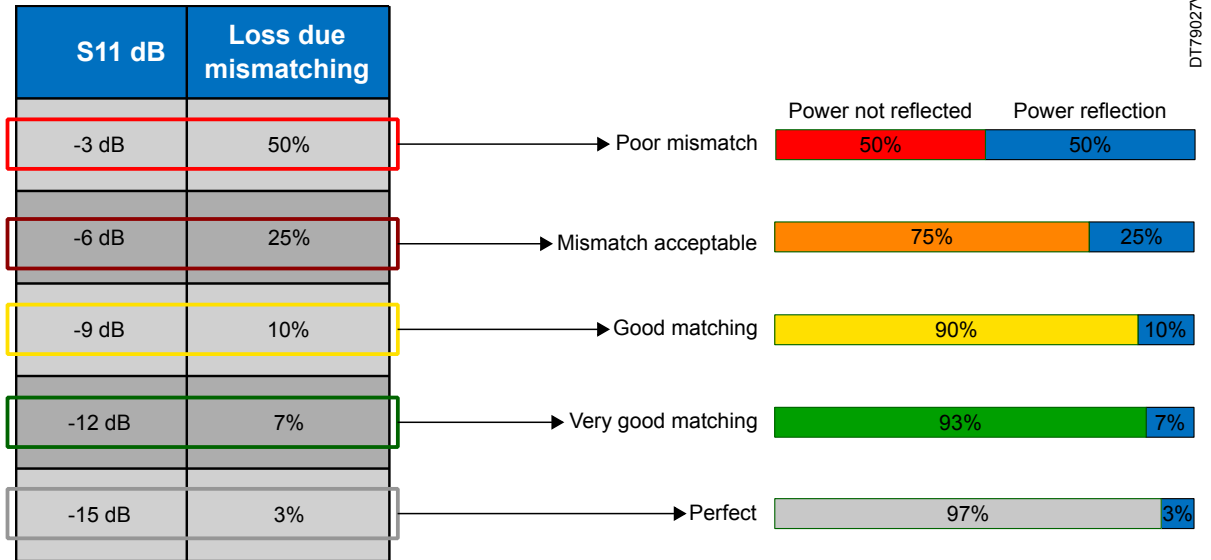
If $S_{11} = 0$ dB then nothing is radiated because all the power is reflected.

With $S_{11} = -10$ dB less power is reflected than with $S_{11} = -3$ dB and more power is radiated.

A lower negative value is considered better than a higher one for S_{11} . A good compromise between a small antenna in module and low return loss could be between -9 and -5 dB.

Below figure summarizes the text:

Figure 7. General view relation of S11 and reflected power



For more information about S parameters refer to [3] to analyze antenna performances.

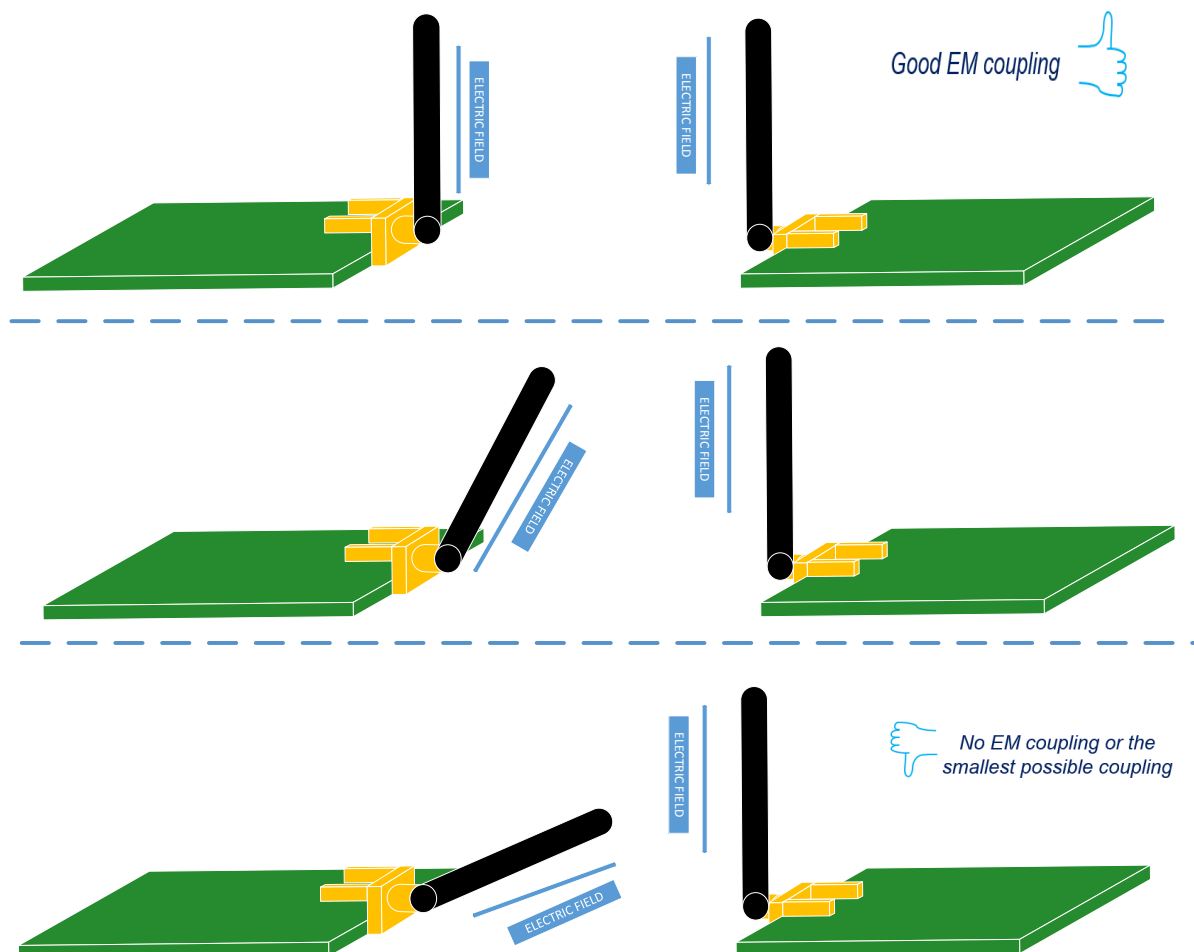
When evaluating the performance of an antenna, it is important to consider also other parameters, not only the S11. The evaluation should also take into account the antenna's polarization, efficiency, and radiation pattern to get a complete understanding of its overall performance.

3.3

Polarization

The polarization of an antenna is the direction of the electric field. A horizontally polarized antenna will not communicate with a vertically polarized antenna. If the transmitting and receiving antennas are rotated from each other, there is a polarization loss that depends on the angle.

Figure 8. Polarization coupling



DT79028V1

3.4 Antenna gain (G)

An antenna is a passive element, and the term “gain” describes how much power is transmitted in the direction of peak radiation compared to a reference isotropic or dipole antenna. Antenna gain is related to directivity and efficiency by:

$$G_{Gain} = \varepsilon_{efficiency} \times D_{directivity}$$

Antenna gain is often specified in terms of dB, dBi or dBd in datasheets:

- dB or dBi: describe the gain relative to the isotropic antenna. A gain of 3 dBi means 2x the power relative to the isotropic antenna in the peak direction.
- dBd: is the decibels relative to a dipole antenna and describes the gain relative to a dipole antenna. The dipole antenna has a gain of 2.15 dBi (compared to the isotropic antenna).
- The relationship between dBi and dBd:

$$dBd = dBi + 2.15$$

3.5 Antenna directivity (D)

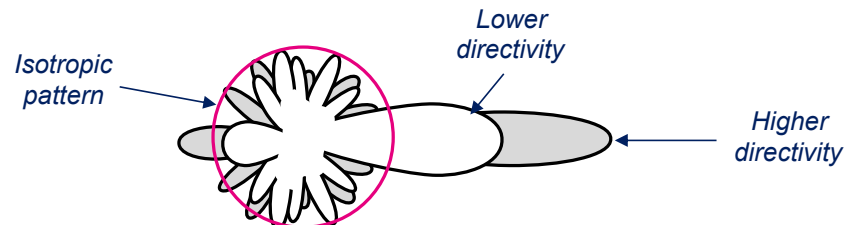
Antenna directivity is a measure of how directional the radiation pattern of the antenna is. An antenna that has the same amount of power radiated in all directions has a directivity of 1 (0 dB) meaning no directionality.

The directivity of an antenna is related to gain and efficiency by:

$$G_{ain} = \varepsilon_{efficiency} \times D_{directivity}$$

The directivity of an antenna is often expressed in dB (decibels).

Figure 9. Directivity view isotropic antenna and directional antenna



DT79029V1

3.6 Antenna efficiency (ε)

Antenna efficiency is the ratio between the power delivered to the antenna (input) and the power radiated by the antenna:

$$\varepsilon = \frac{P_{radiated}}{P_{input}}$$

The antenna efficiency is a value between 0 and 1. It can also be expressed in decibels (dB).

Example

If an antenna has an efficiency of -3 dB, it means that only ~ 0.5 or 50% of the power delivered to the antenna is radiated.

In terms of gain (in dB):

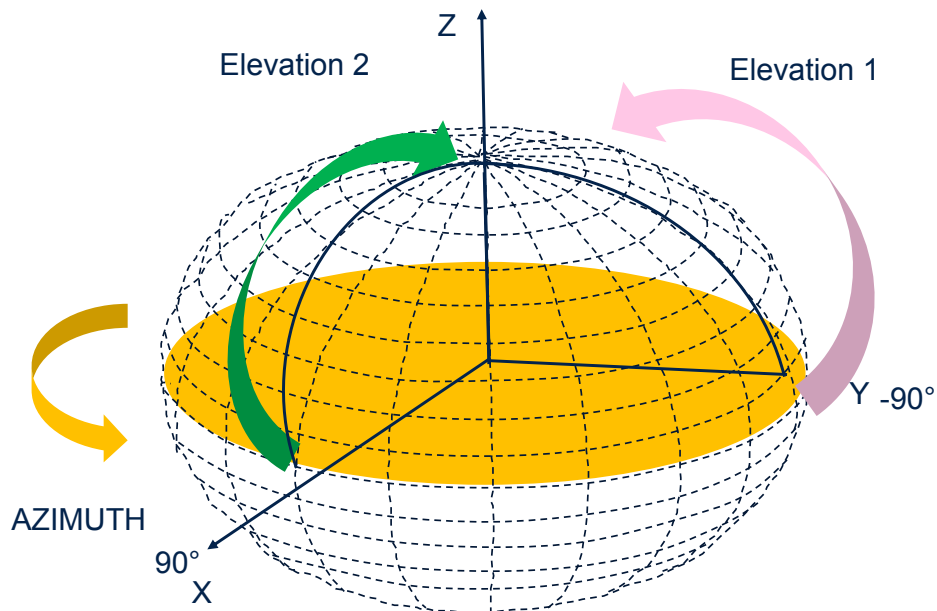
$$G_{dB} = \varepsilon_{dB} + D_{dB} = -3 + D_{dB}$$

So the antenna gain is reduced by 3 dB when its efficiency is 50% (compared to a 100% of efficiency).

3.7 Antenna radiation pattern

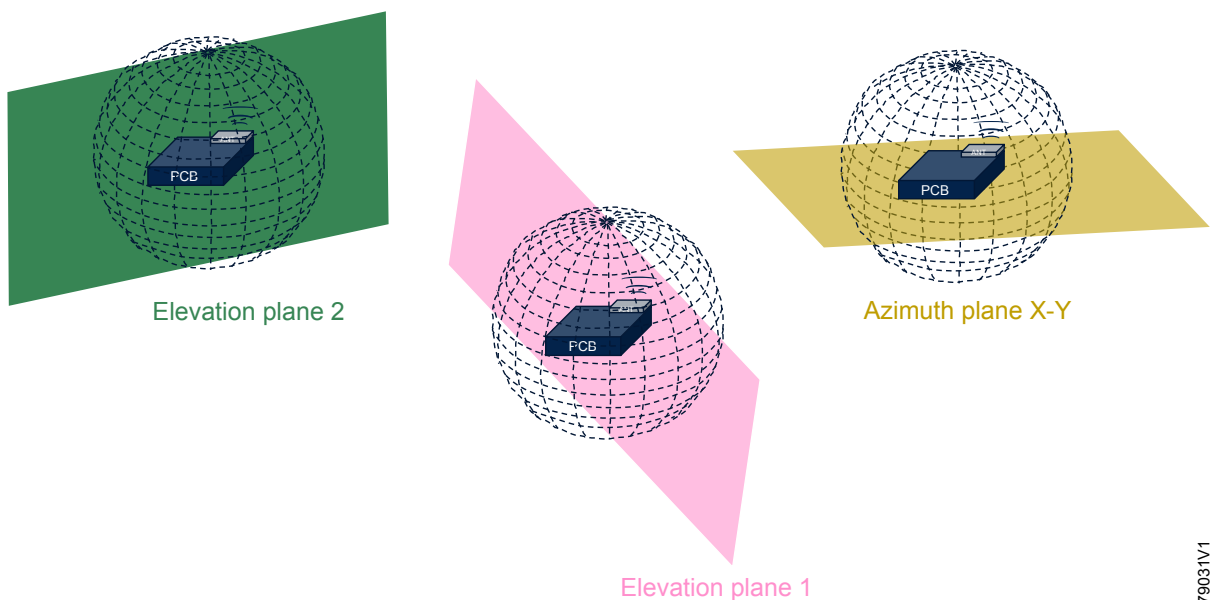
To have an idea of how the antenna radiation is distributed in the space, 2D polar plots are used as a projection on a plane of the 3D radiation pattern.

Figure 10. Radiation pattern plane view



DT79030V1

Figure 11. Radiation pattern planes view 2

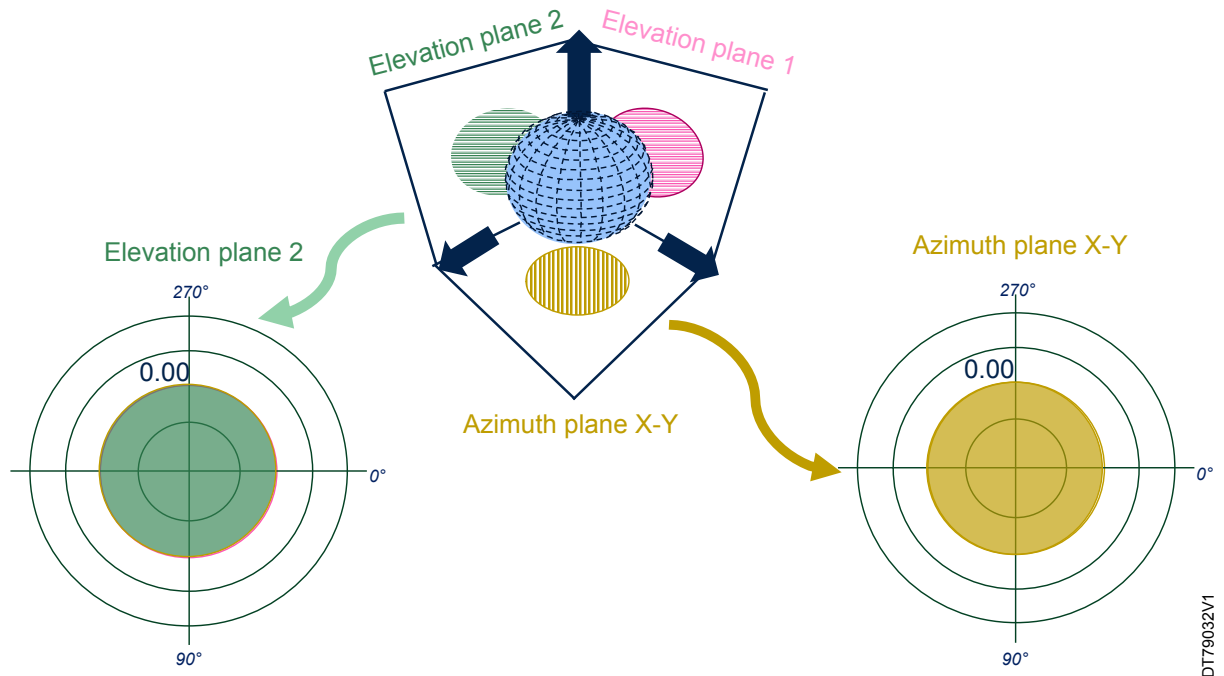


DT79031V1

The radiation pattern plot is useful for identifying the zone of maximum gain of an antenna. This maximum gain is compared to an isotropic antenna or a dipole antenna.

The isotropic antenna is an ideal antenna (it does not exist in practice) that radiates the same amount of power in all directions:

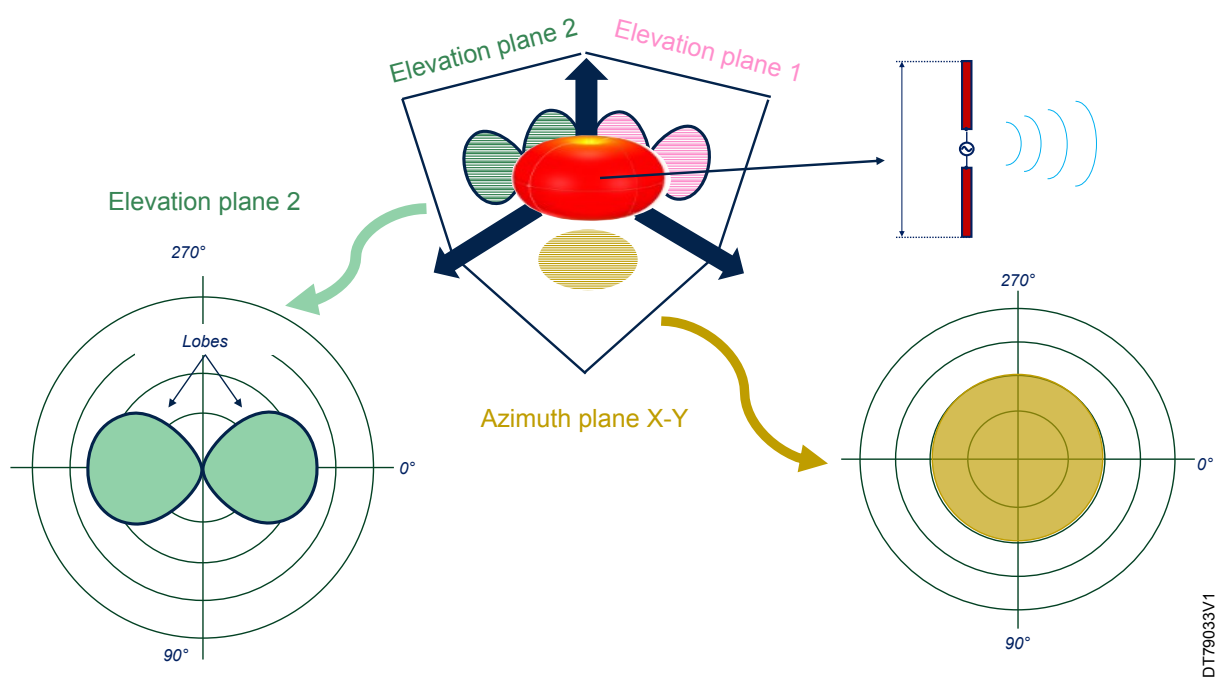
Figure 12. Radiation pattern antenna isotropic



DT79032V1

The same comparison is made with a dipole antenna, a classical antenna that has a donut-shaped radiation pattern:

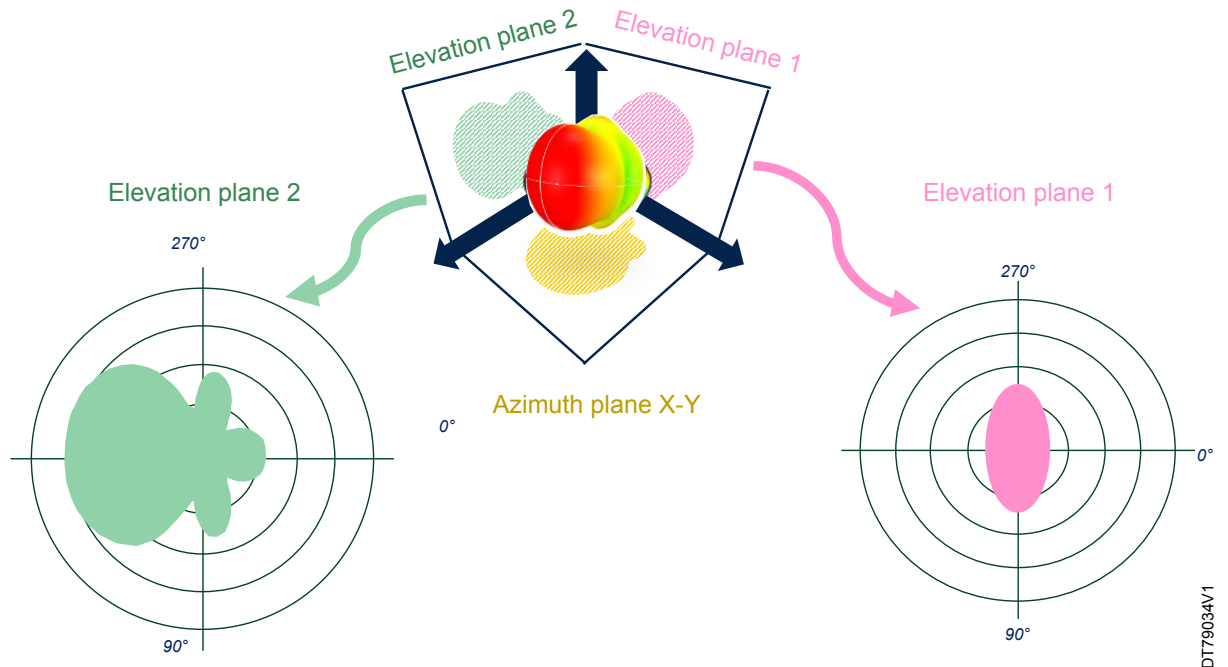
Figure 13. Radiation pattern antenna dipole



DT79033V1

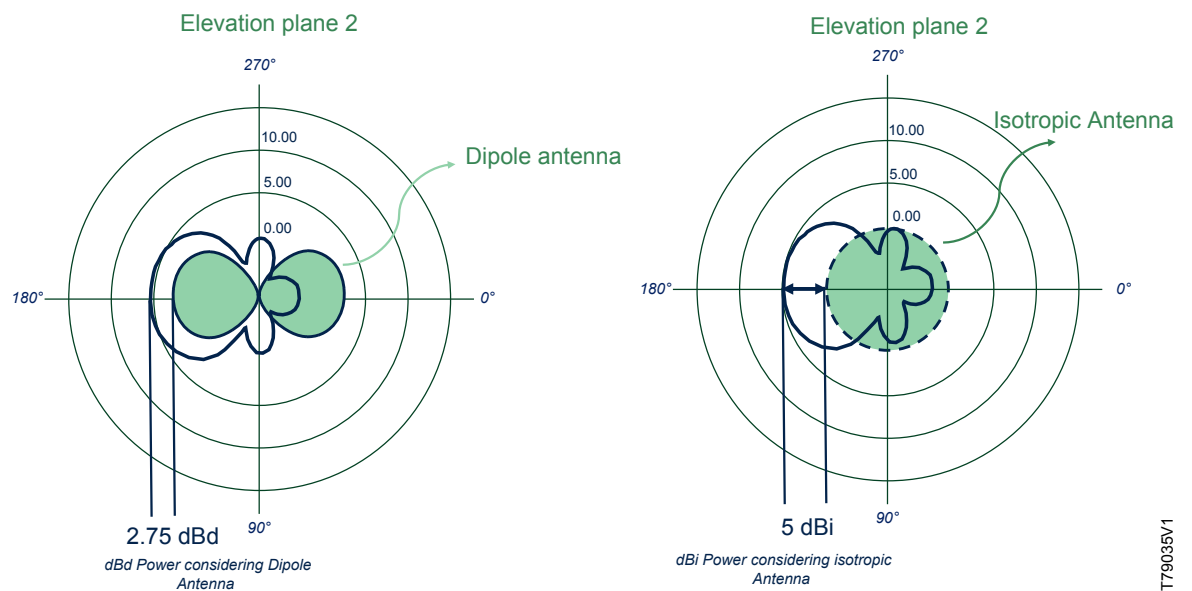
An antenna with the radiation pattern shown below is considered; this is called antenna Vivaldi, for instance:

Figure 14. Radiation pattern antenna Vivaldi



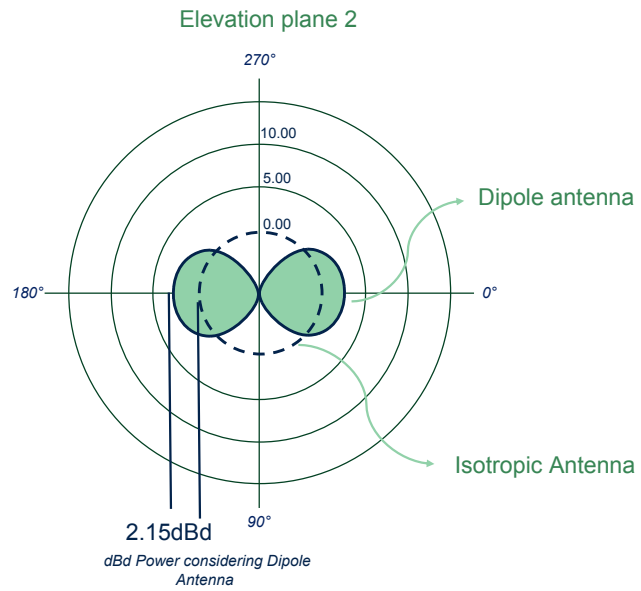
The maximum gain of this antenna, considering a dipole antenna or an isotropic antenna, can be seen in the figure below:

Figure 15. Dbi and Dbd gain example for antenna Vivaldi



The formula for converting dBd to dBi can still be derived considering the figure below:

Figure 16. Dbi and Dbd relation



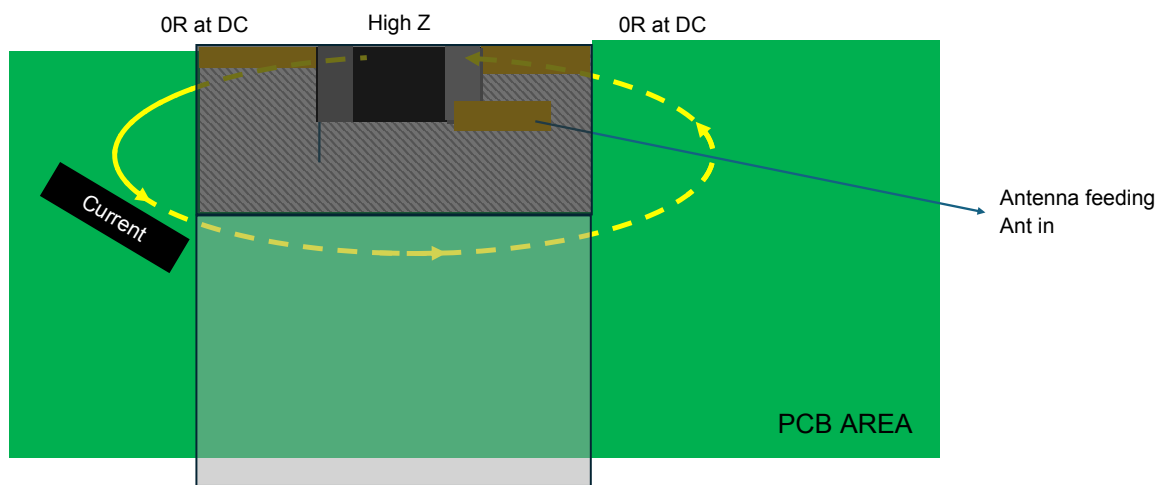
DT79036V1

4 Module internal antenna

The STM32WBA5MMG uses an internal ceramic antenna with a slot-type topology. This antenna is very small and relies on a ground plane in the PCB to radiate effectively. The slot antenna type is considered an LC resonator. The capacitance (C) is defined by the ceramic antenna component, and the inductance (L) is proportional to the length of the current path around the antenna clearance on the PCB. The resonance frequency is inversely proportional to LC. Increasing either C or L inadvertently decreases the frequency. The layout must be designed with particular care to avoid performance loss caused by a shift in LC resonance.

In the figure below, there is a general view of the internal antenna in the module and the current path around the clearance area:

Figure 17. Module internal antenna and current path



To illustrate the impact of layout on antenna performance, Figure 18 shows how variations of the clearance area impact the quality of the antenna design. The optimal design enables the antenna to operate with minimal loss across the frequency range in the desired band.

A larger clearance area results in a longer current path and reduces the resonance frequency. If the clearance area is too small, the frequency increases. There is an opportunity to fine-tune the resonance frequency by adjusting the nearly optimal clearance area.

Below is the effect of a change in clearance and S11 shift:

Figure 18. Relation between antenna clearance and S11

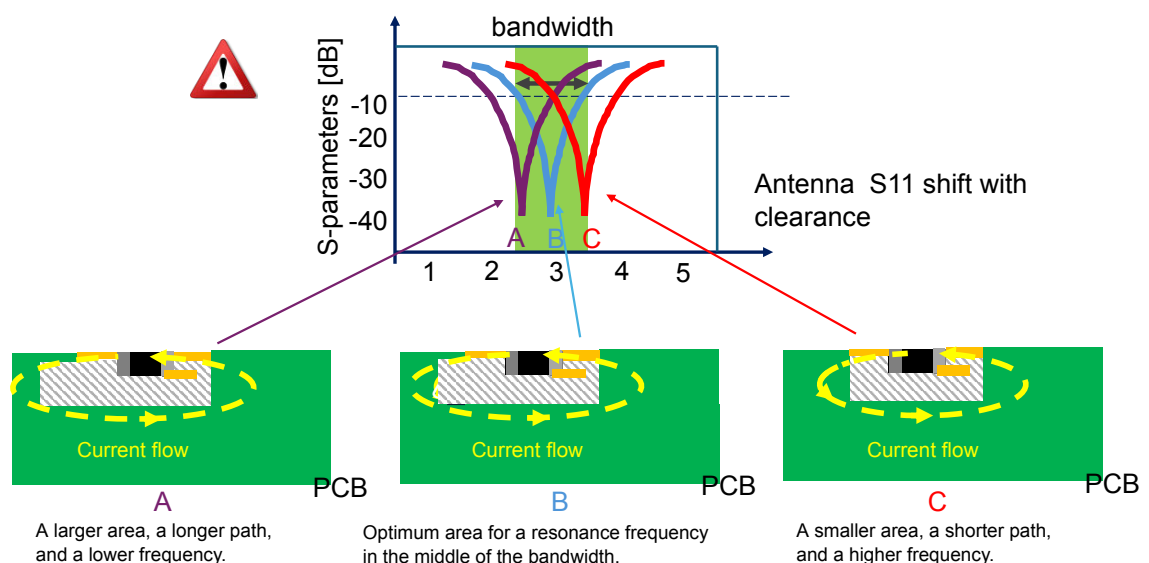
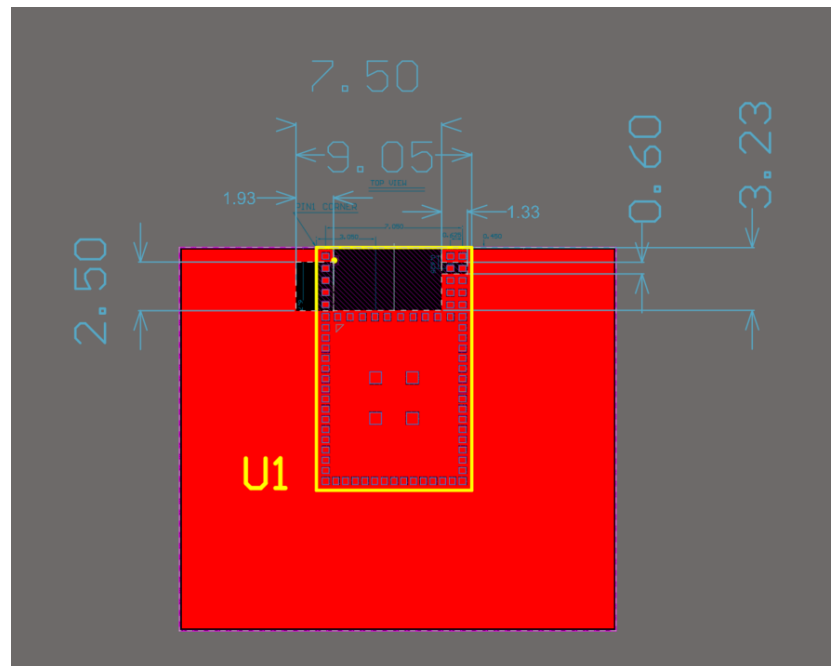


Figure 18 shows that when the clearance is increased, the S11 parameter shifts lower, and when the clearance is decreased, it shifts higher. Three scenarios are depicted in the image to demonstrate this effect. Changes in clearance affect the current path, altering inductance and shifting the LC resonance. Customers may face various constraints related to PCB dimensions, board stack-up, plastic housing, and module proximity to the body. Guidelines are provided for easy fine-tuning. Customers can choose between using a reference design or customizing their design for greater flexibility, which is recommended.

Option 1

Use reference design layout like clearance in the Figure 19. For reference design refer to [4].

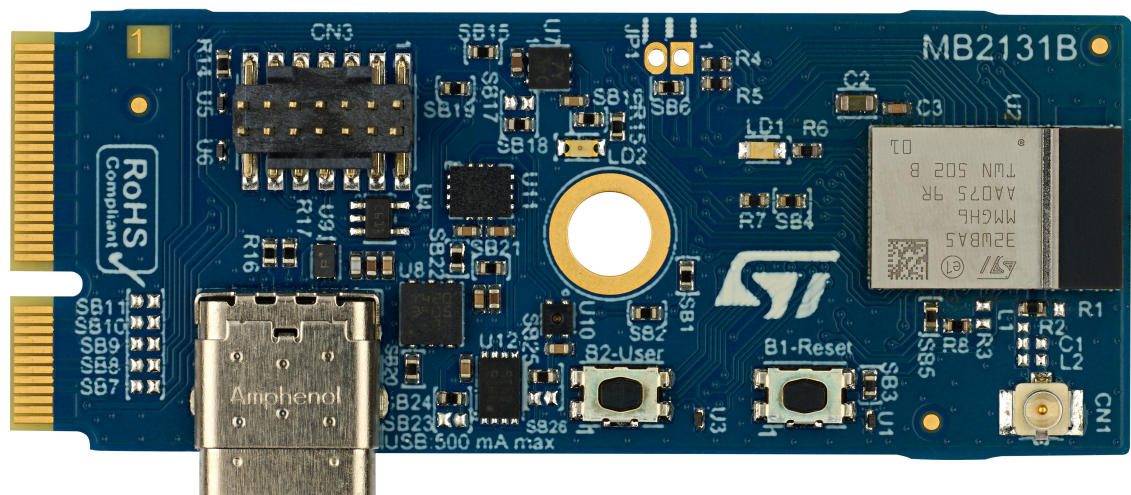
Figure 19. Clearance for reference boards



DT79039V1

Note: All dimensions in the above layout and in the following ones are in millimeters (mm).

Figure 20. Connectivity expansion board (CEB)



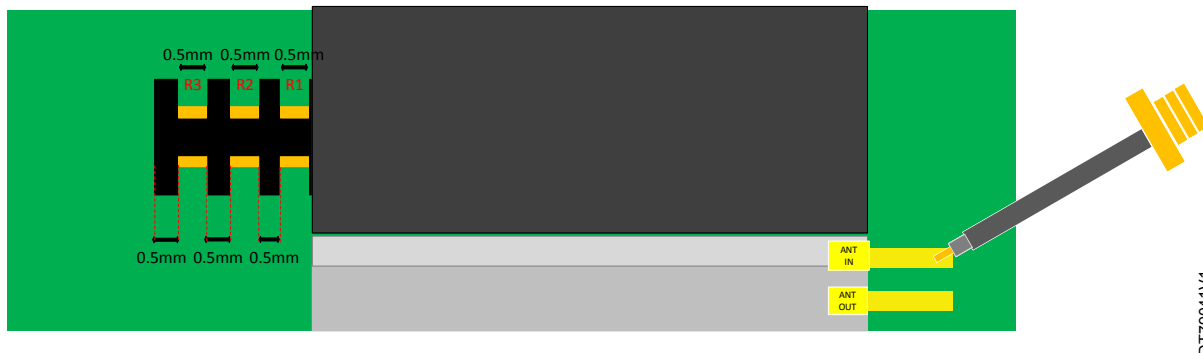
DT79040V1

For CEB files and layout footprint refer to [4] in case customer needs reference design.

Option 2: fine-tuning

To provide customers with more flexibility, fine-tuning is recommended. The main idea is to test three different configurations, which involve possible connections of resistors spaced by 0.5 mm. This approach aims to shift the S11 parameter by adjusting the clearance, as explained in Figure 18:

Figure 21. Option 2: fine-tuning

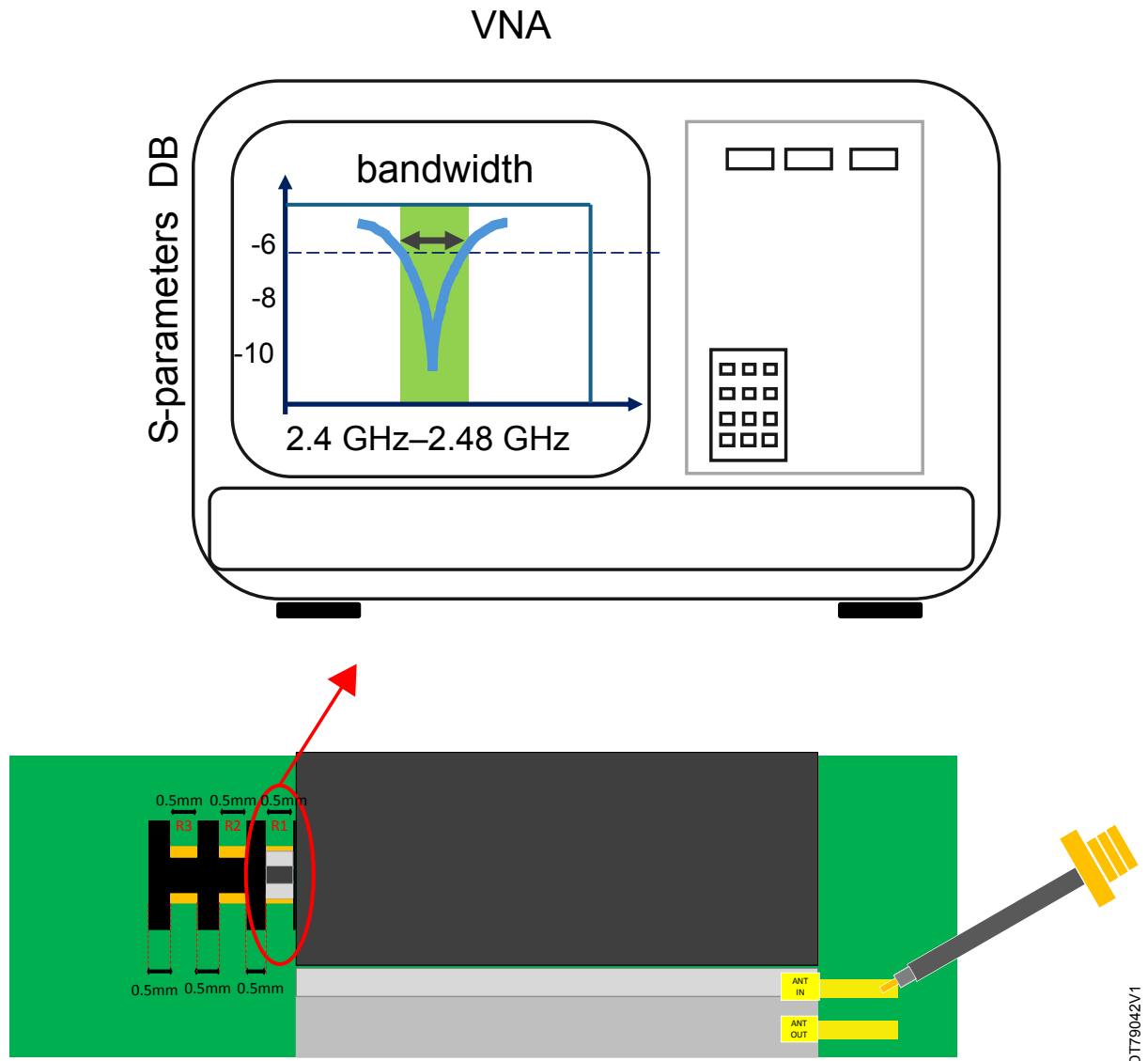


The steps for fine-tuning are explained below:

Step 1

Measure S11 with 0 Ω as R1 resistance soldered in the PCB using a probe test in VNA (network analyzer).

Figure 22. Step 1 fine-tuning

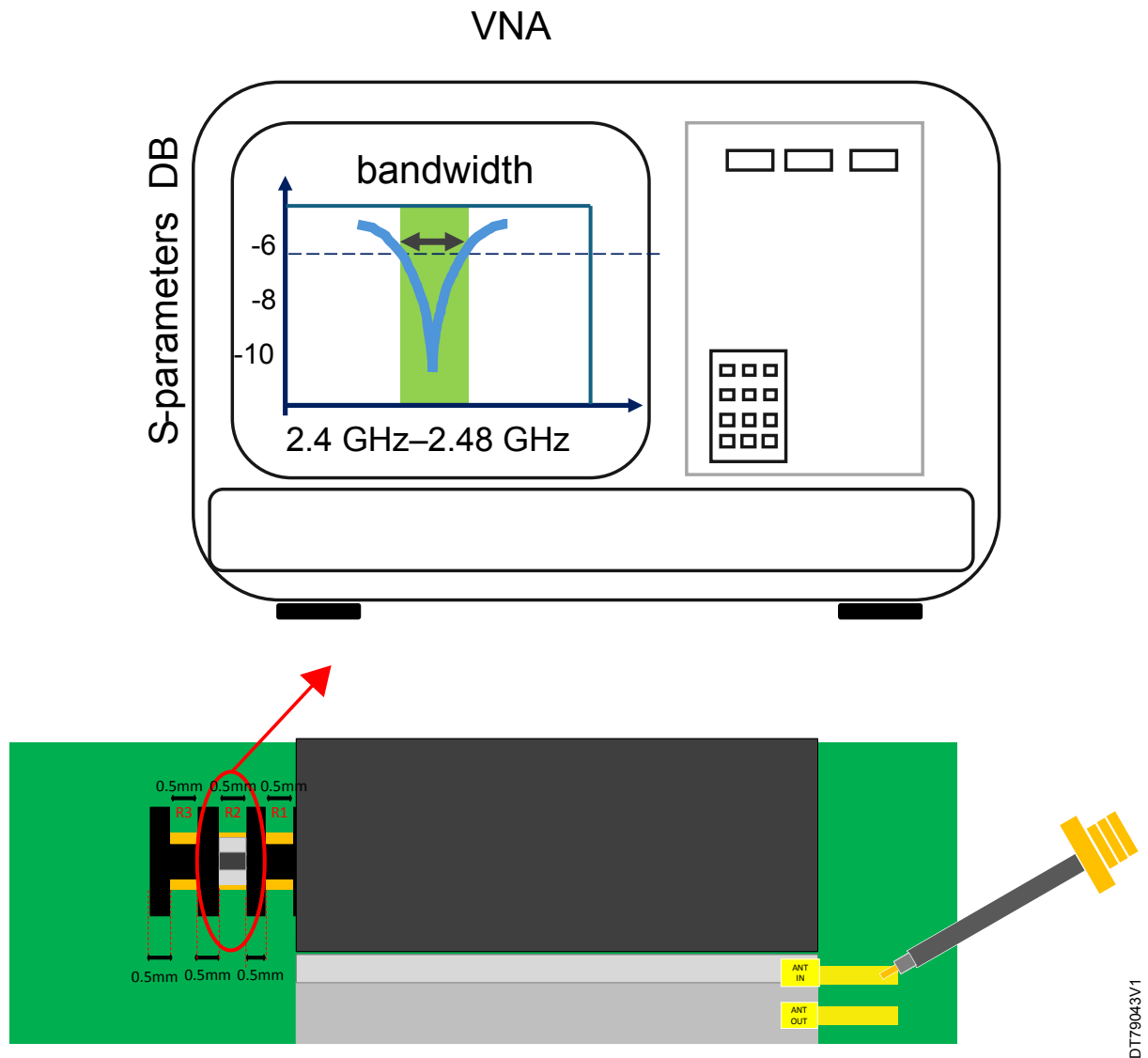


If S11 has the correct margin in 2.4 GHz band, no further steps are needed. Otherwise, proceed with step 2.

Step 2

Measure the S11 parameter with a 0 Ω resistor soldered as R2 on the PCB using a probe test with a vector network analyzer (VNA).

Figure 23. Step 2 fine-tuning

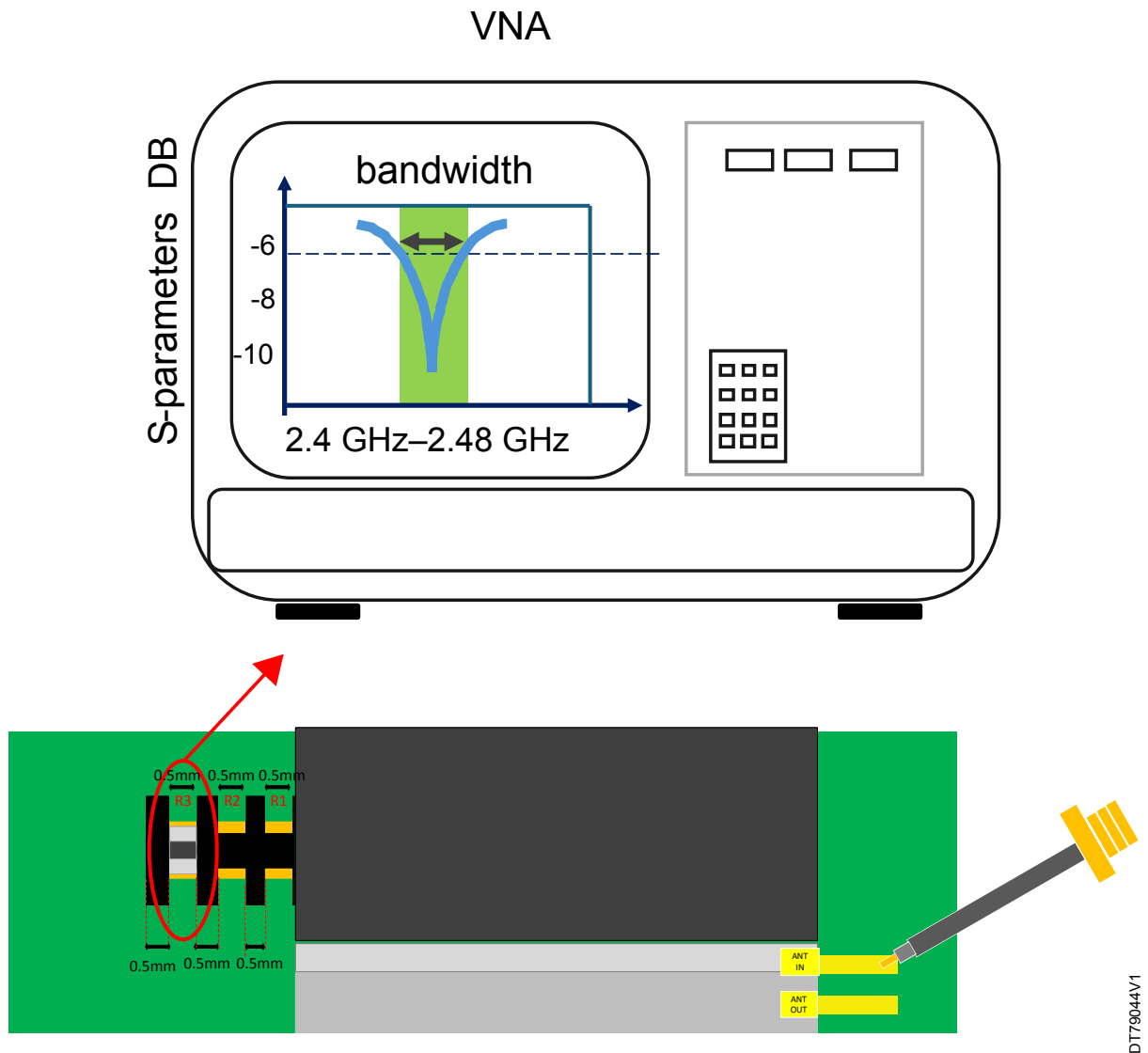


If S11 has correct margin in 2.4 GHz band, no more steps are needed otherwise proceed with step 3.

Step 3

Measure with a 0 Ω resistor soldered as R2 on the PCB using a probe test with a vector network analyzer (VNA).

Figure 24. Step 3 fine-tuning

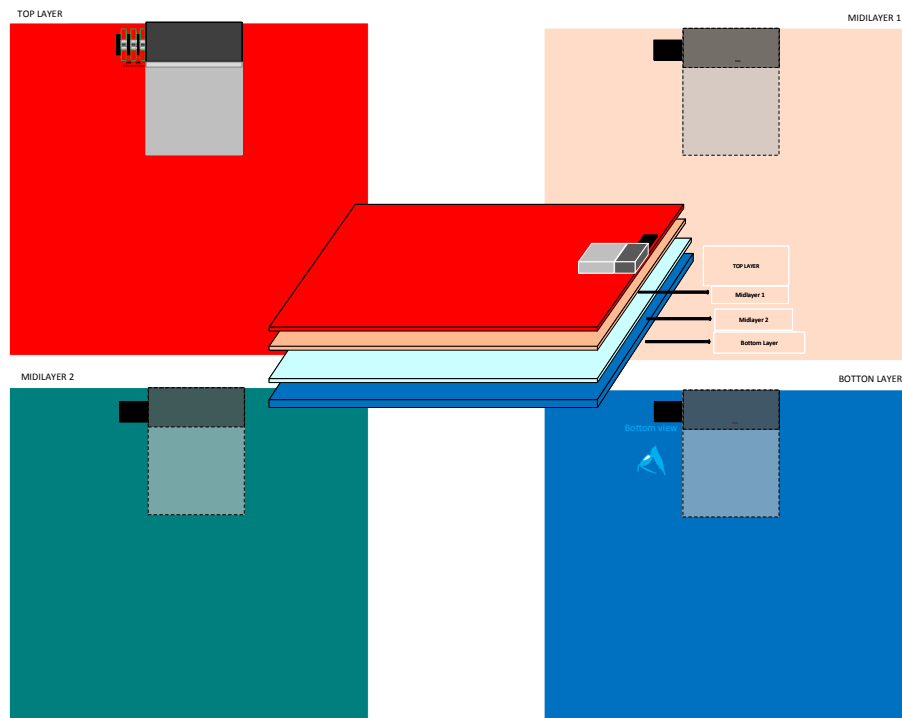


Inserting R3 S11 should have correct margin in 2.4 GHz band.

In the end, the customer should choose the resistance configuration that gives the best results.

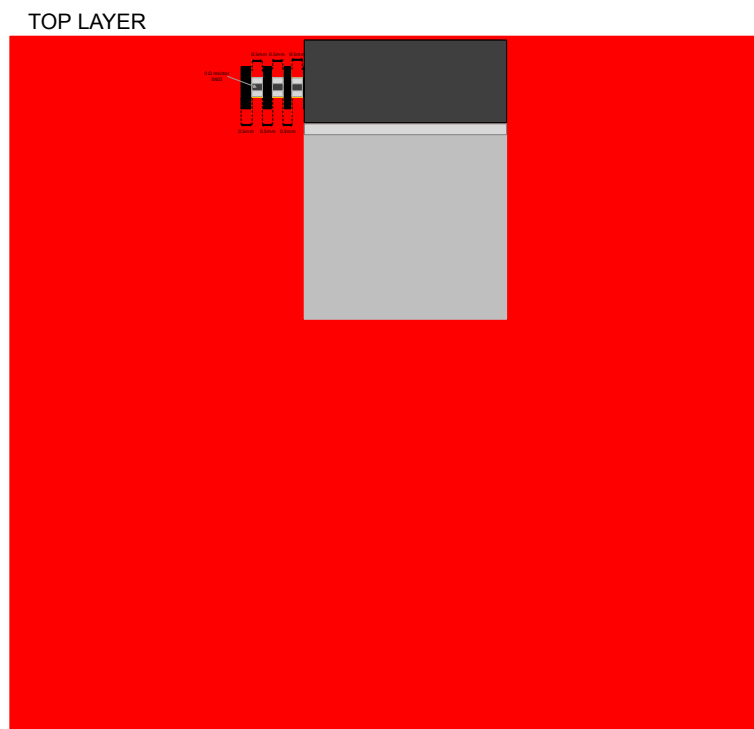
A global view of the 4-layer board with fine-tuning possibilities is provided below:

Figure 25. 4 layer board global view tuning



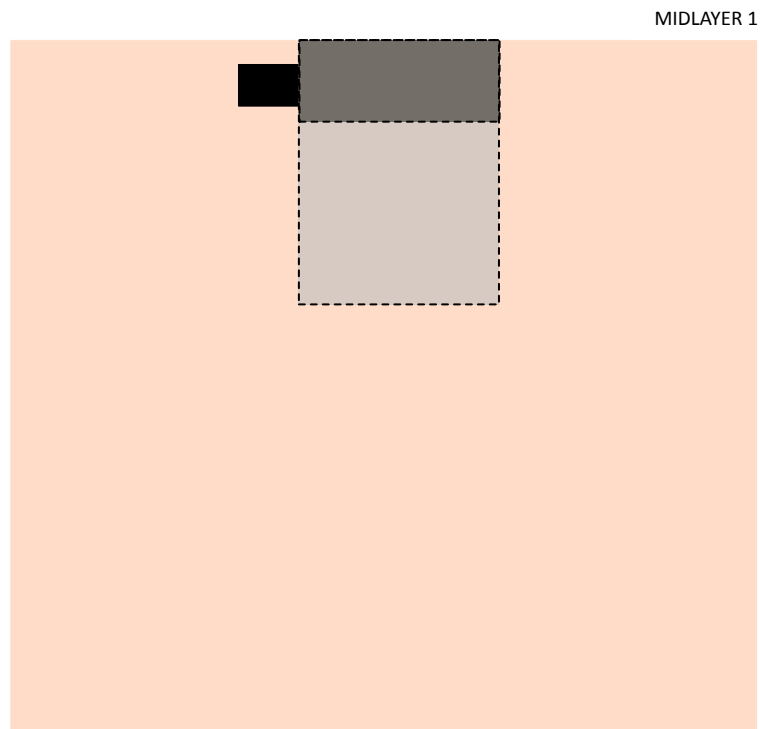
DT79045V1

Figure 26. Top layer guidelines



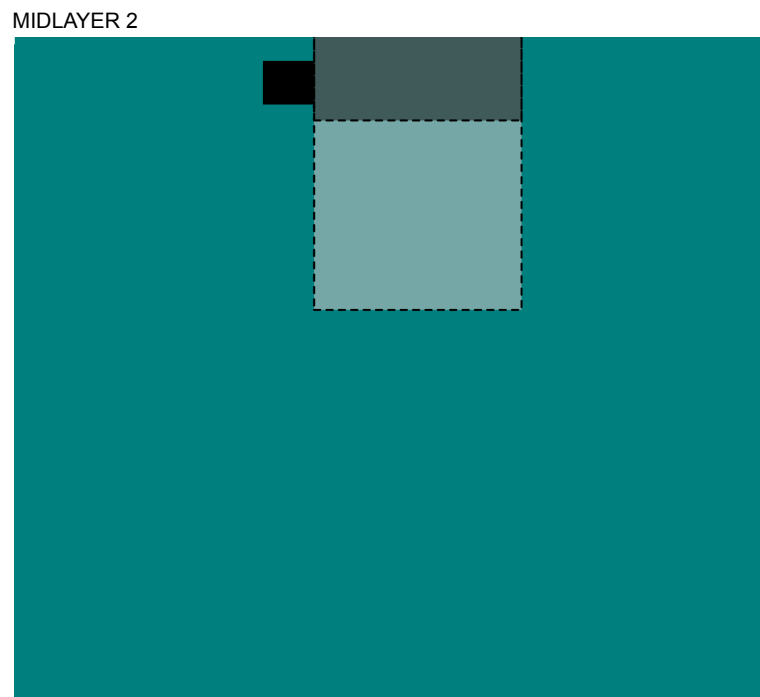
DT79046V1

Figure 27. Midlayer 1



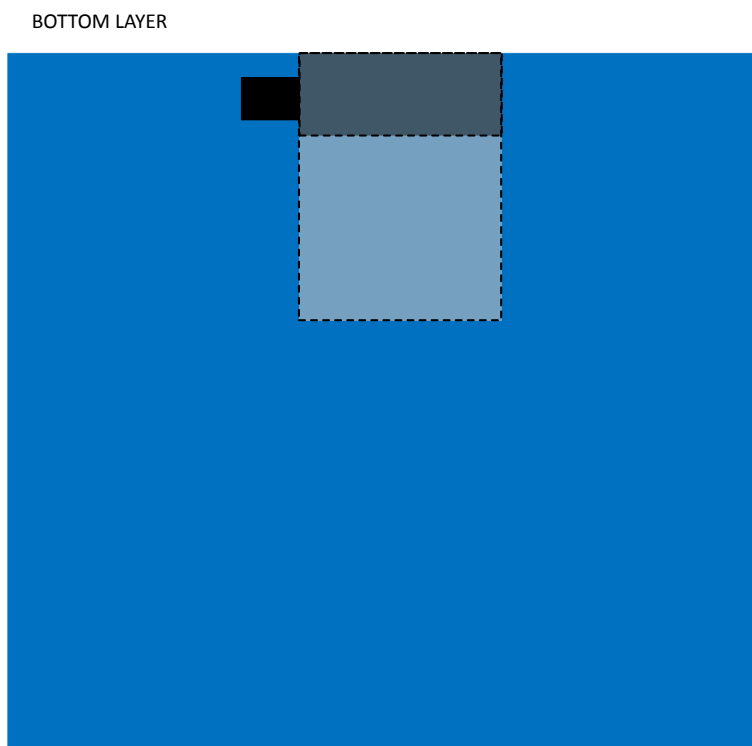
DT79047V1

Figure 28. Midlayer 1 guidelines



DT79048V1

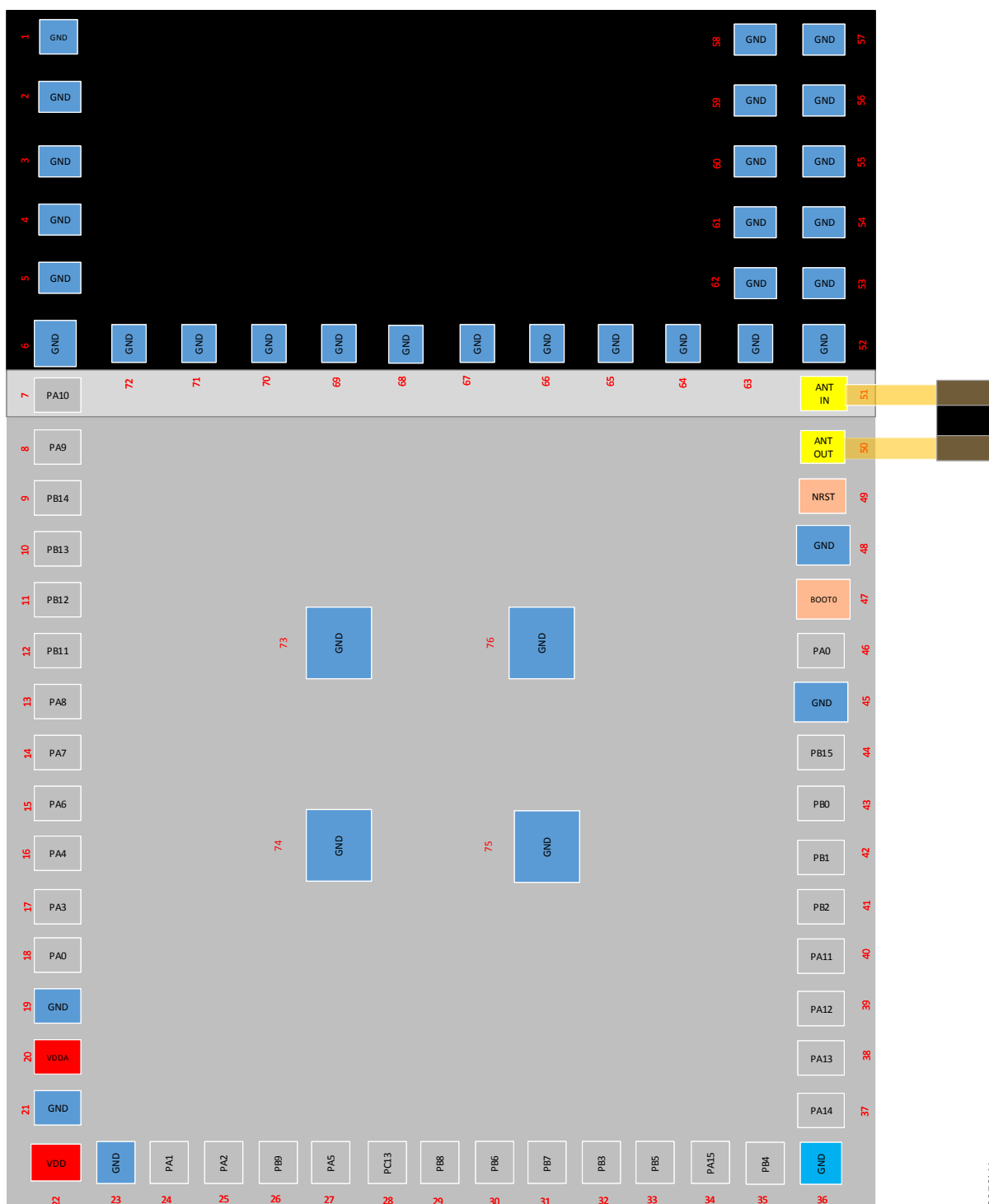
Figure 29. Bottom layer



DT79049V1

To fine-tune the module, it is mandatory to have access to the ANT_IN pin. It is recommended to connect ANT_IN and ANT_OUT with a 0 Ω resistor and place it is very close to these pins. For the tuning this resistor can be removed and a probe connected to ANT_IN.

Figure 30. Connection between antenna out and antenna in



DT79051V1

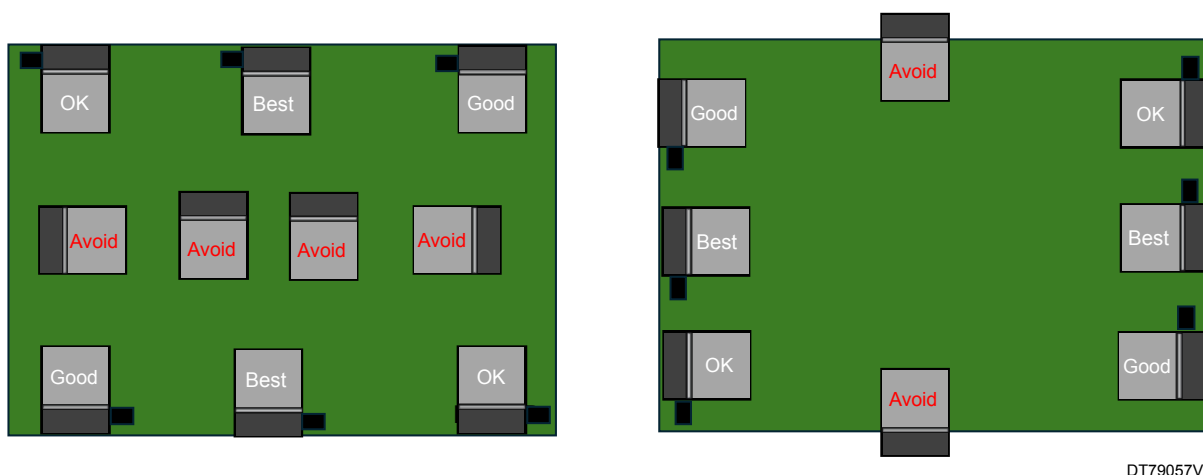


5 Module placement

This section provides guidelines for the placement of the STM32WBA5MMG module on a customer board. The module is designed to achieve good RF performance across various placements and board stack-ups, but guidelines should be considered to maximize the module's potential.

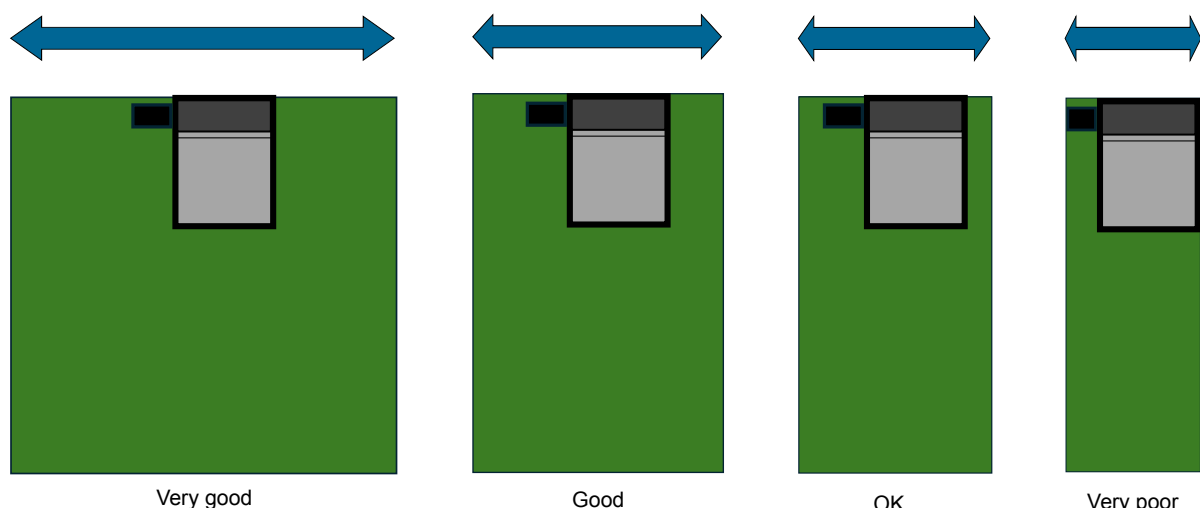
The Figure 32 shows some possible positions of the module with respect to the PCB. The antenna edge of the module should always be aligned with the edge of the PCB. The module should not be placed anywhere in the center of the PCB and it should not stick out. The best placement of the module is in the middle-edge of the PCB, but good performance can be achieved right side and left side if customer has liberty, its always indicated to put the module first in middle after right side and if no choice left side.

Figure 32. Module best placement in board



Note: The performance of the RF module is linked to the dimensions of the PCB in the clearance axis and the respect to clearance. Be aware that any obstruction of this path significantly affects RF performance.

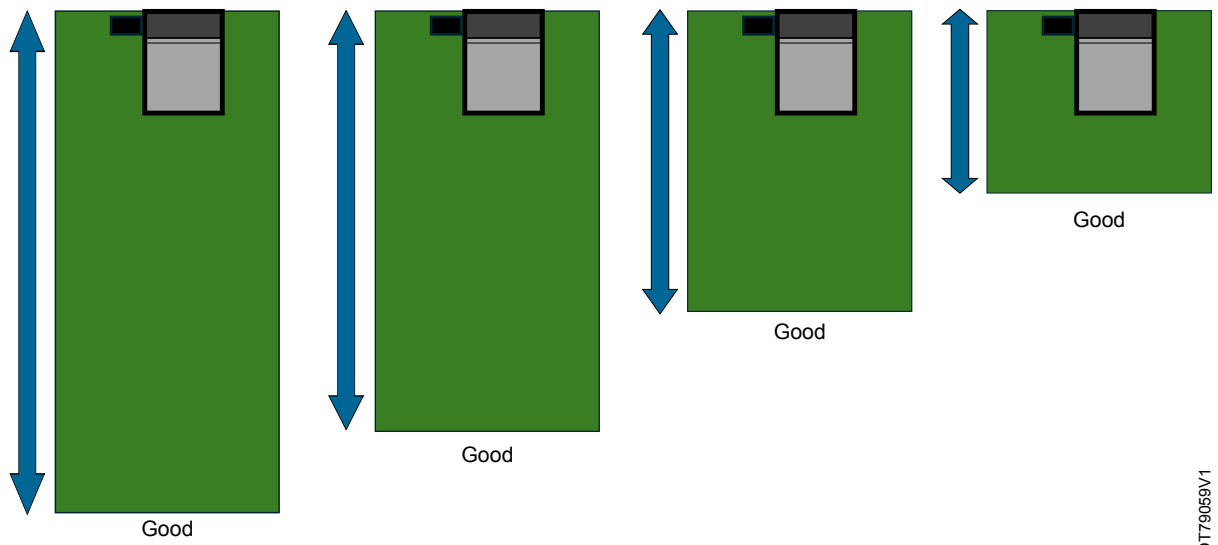
Figure 33. Module performance view with bigger board X axis (clearance axis)



DT79058V1

Note: The performance of the RF module is not critical for the Y dimensions, as this has minimal impact on the antenna performance.

Figure 34. Module performance view with bigger boards Y axis



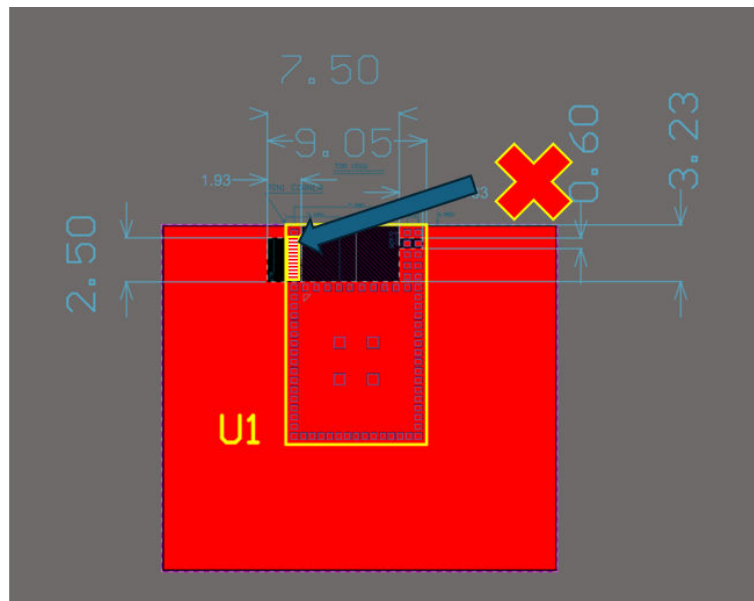
DT79059V1

6 Layout mistakes to avoid

Some important advice about layout clearance: the customer should avoid making the mistakes below because they detune the antenna and compromise the performance for the application.

As a general rule, ground connections (marked GND) should be grounded. However, there are some exceptions: pads #2, #3, #4, #5, #56, and #59 should be left unconnected.

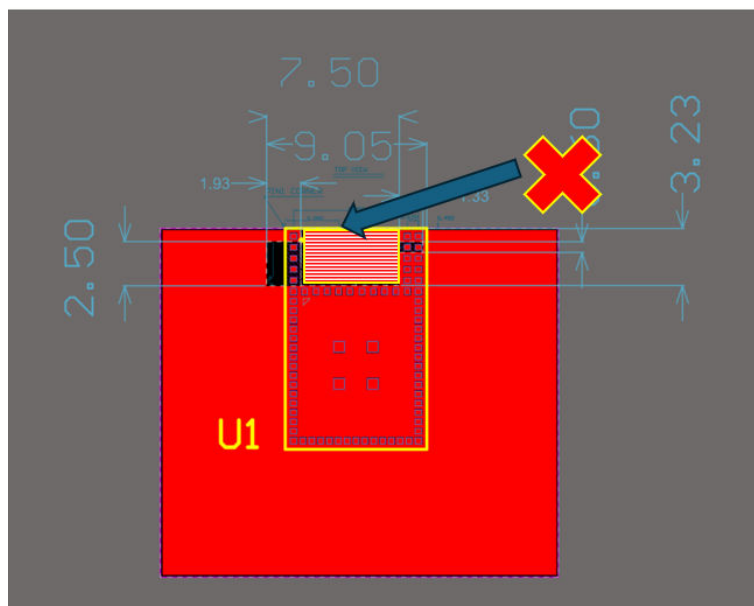
Figure 35. Error 1: do not connect NC pads on the left



DT79052V1

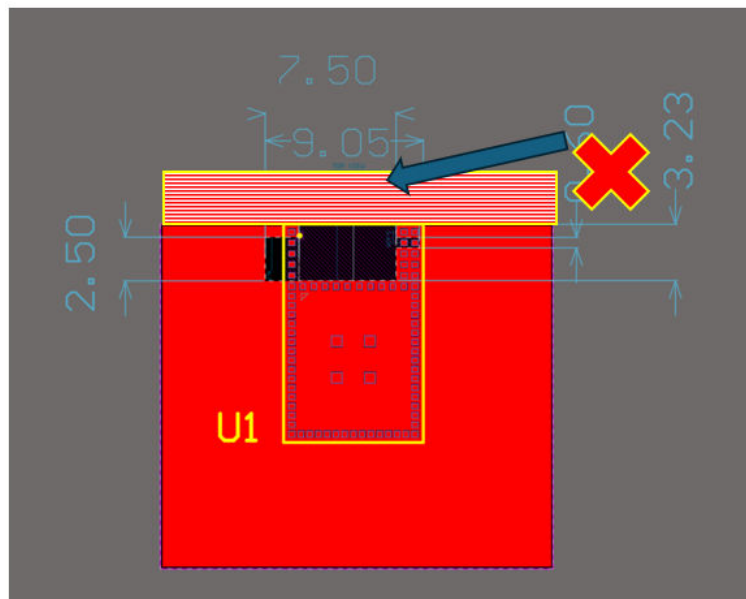
Pads #2, #3, #4, #5 should be not connected to GND neither between them, as they are just mecanical pads.

Figure 36. Error 2: do not place GND in antenna clearance



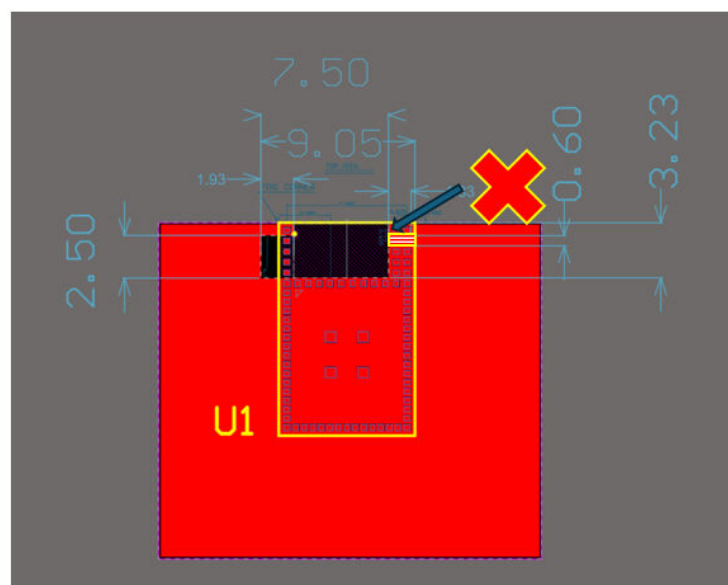
DT79053V1

Figure 37. Error 3: do not place GND above antenna on the board



DT79054V1

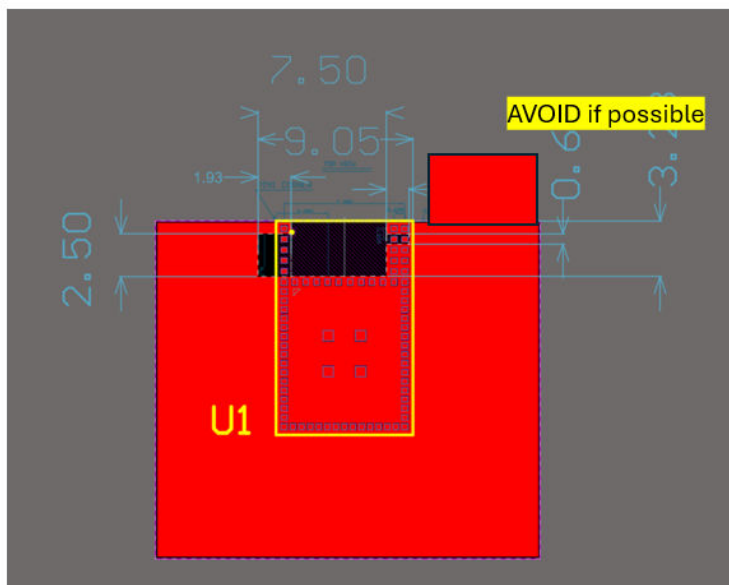
Figure 38. Error 4: do not connect GND to NC pads on the right



DT79055V1

Pads #56 and #59 should not be connected to GND or to each other, as they are just mechanical pads.

Figure 39. Error 4: avoid GND below on the right, if possible

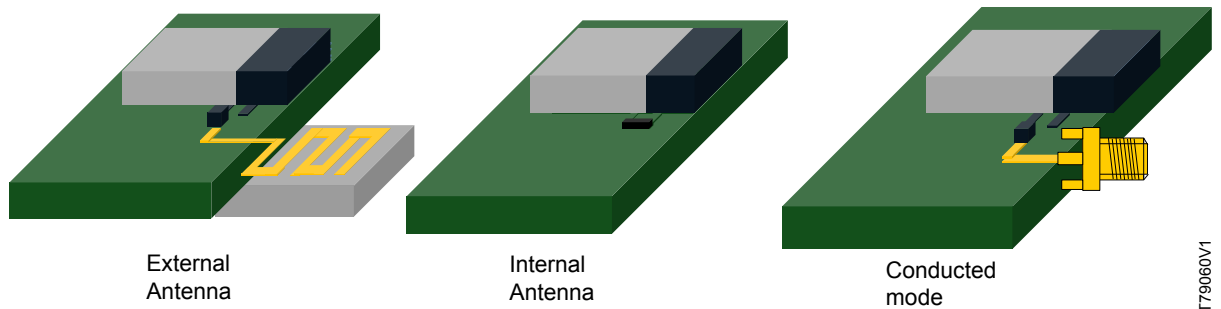


DT79056V1

7 External antenna

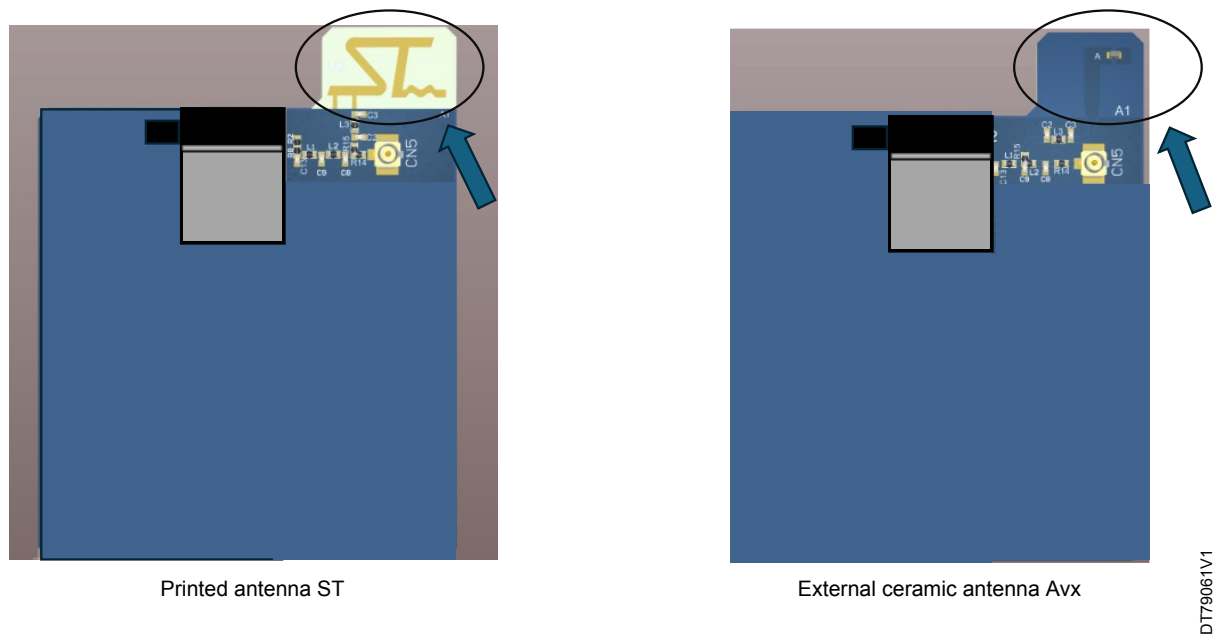
The module can be used with its internal antenna. Alternatively, it is possible to place a PCB or ceramic antenna on the board, or add a UFL or SMA connector:

Figure 40. Module RF configurations



The customer has the option to use an external antenna instead of the embedded antenna. In this case, two reference designs are suggested:

Figure 41. Module designs with external antenna



8 Schematics

For operation with the internal ceramic antenna, the loop from ANT_IN to ANT_OUT should be closed with an R0 resistor. Ideally, the resistor is placed close to and parallel with the module edge to keep the loop as short as possible. If the device needs flexibility for a connection to an external antenna or test equipment in conducted mode, then connect ANT_IN to GND, and the connection to ANT_OUT can be made through a perpendicular resistor.

Figure 42. Antenna in and antenna out pin view

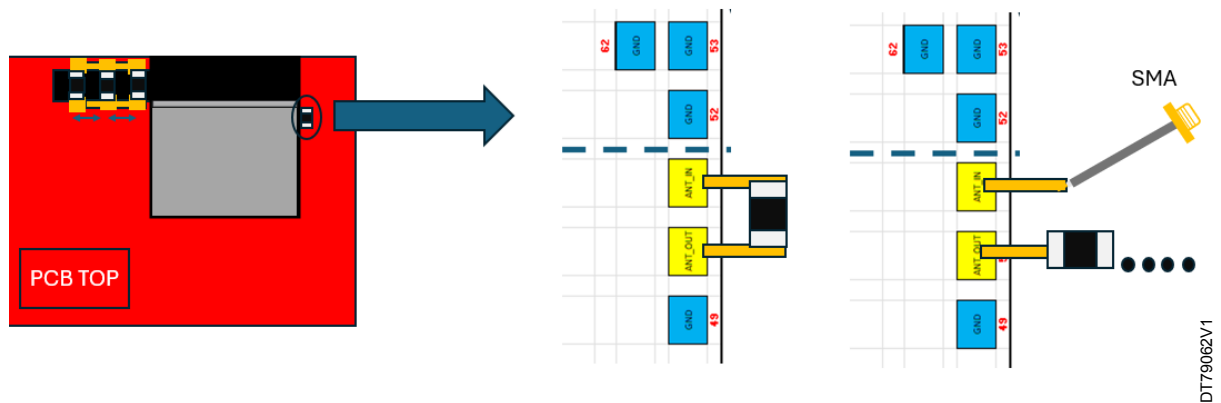
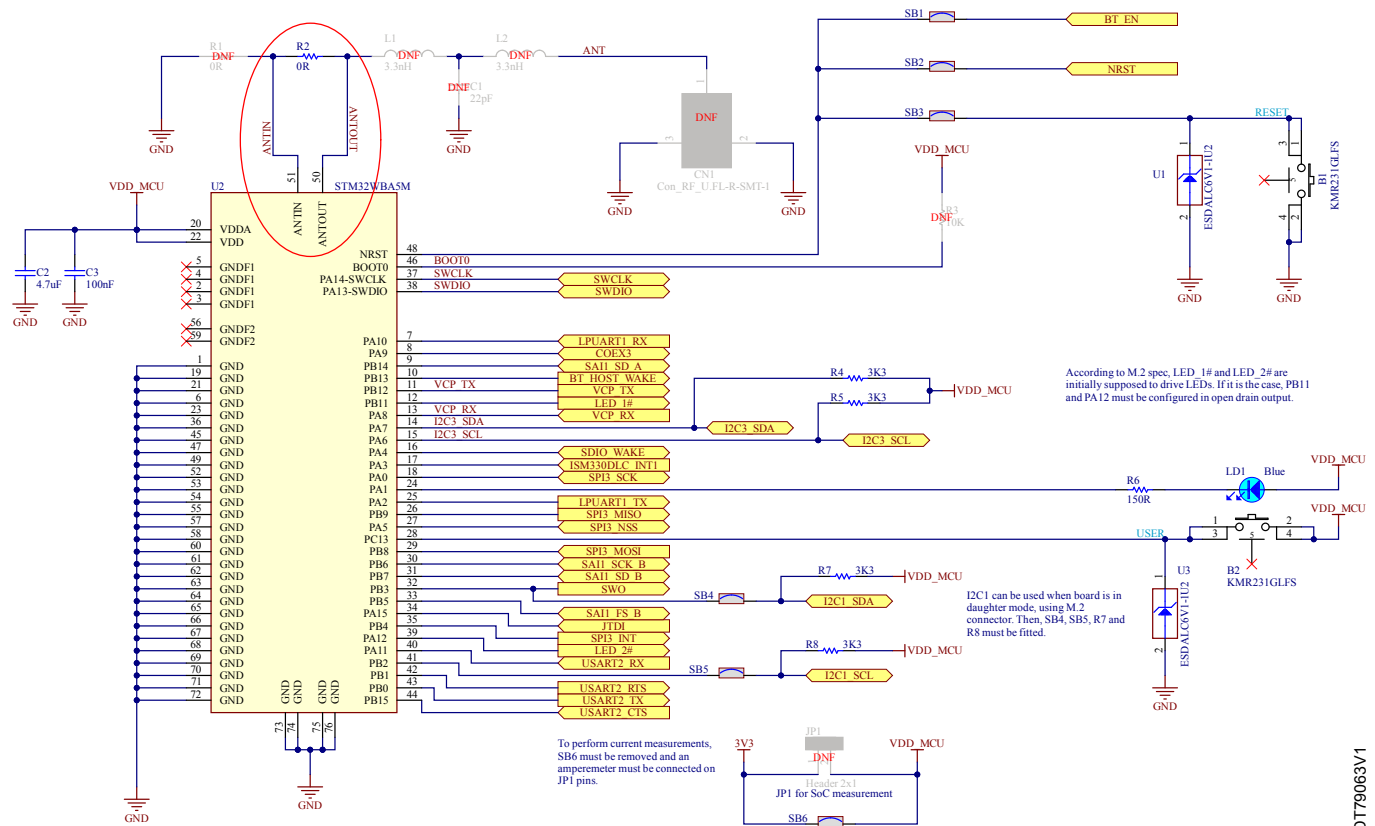


Figure 43. CEB schematics module part



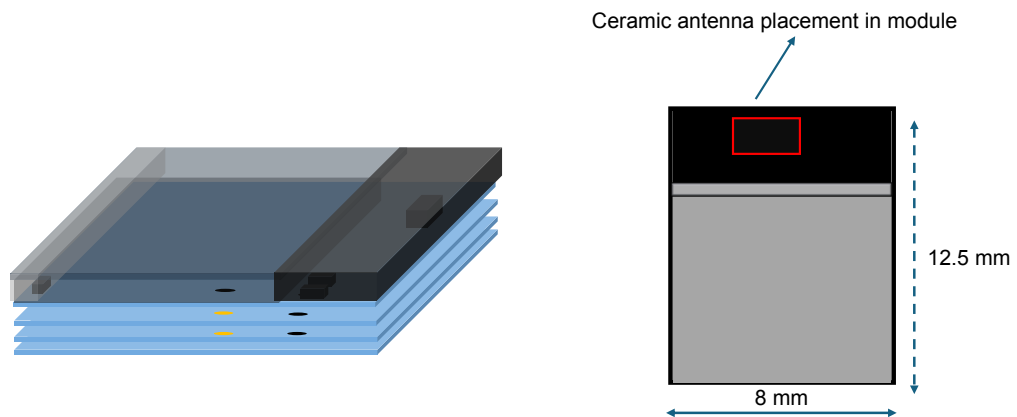
For schematics of CEB and layout files refer to [4].

9 Antenna supplementary constraints

As usual, the antenna is very sensitive to any nearby materials, such as plastic, the human body, and especially metal objects. This section provides some insights about the effect of close objects on RF performance.

The antenna position in the module is like below:

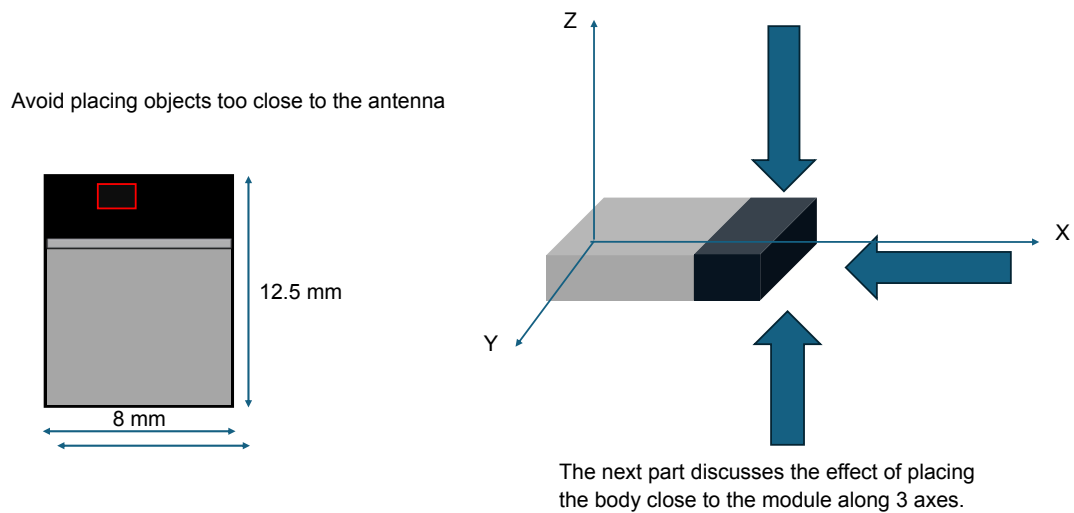
Figure 44. Module antenna placement view



DT79064V1

Even though molding provides some robustness to nearby materials for the antenna, it is advised to maintain a distance from the antenna region, as any nearby material can detune the antenna to some extent:

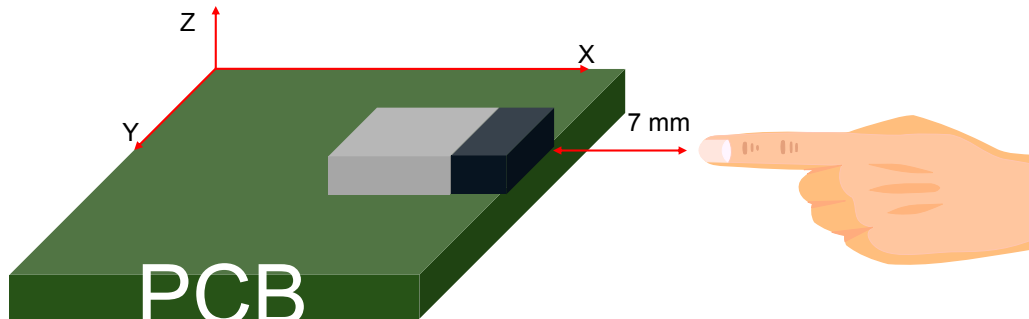
Figure 45. Approach of external object toward module along main axes



DT79065V1

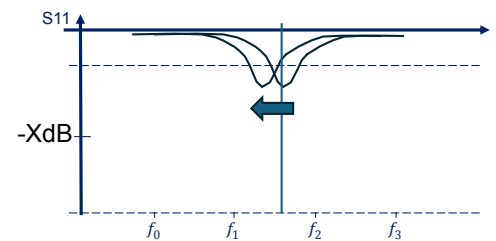
Body influence along the X axis:

Figure 46. Effect of body approach toward module along X+ axis S11



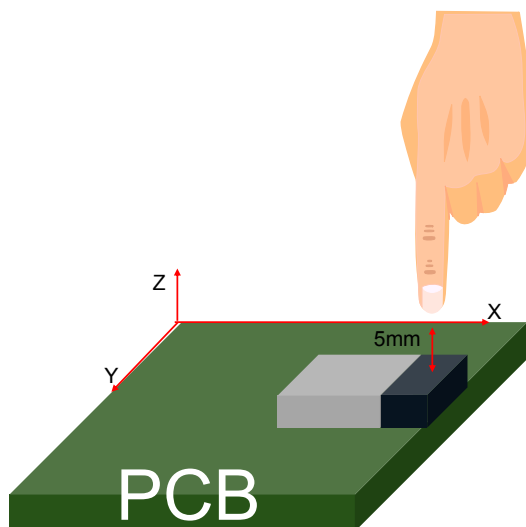
DT79066V1

Approach body like the figure shows the effect of push matching to left. It is recommended to have a space of ideally 7 mm or more to reduce the impact of detuning antenna matching.

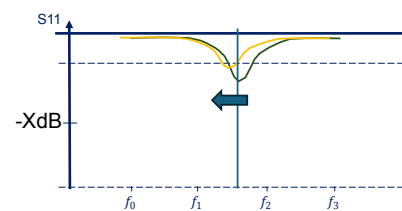


Body influence in Z up:

Figure 47. Effect of body approach toward module along Z+ axis S11



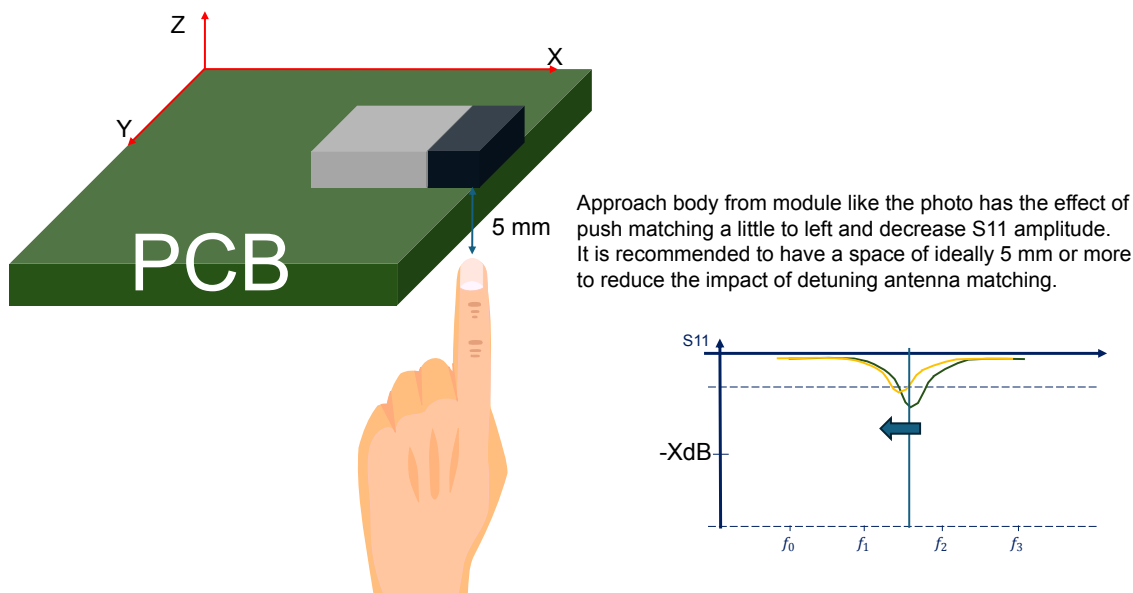
Approaching the body to the module like in the picture has pushing effect to matching a little to left and decreasing S11 amplitude. It is recommended to have a space of ideally 5 mm or more to reduce the impact of detuning antenna matching.



DT79067V1

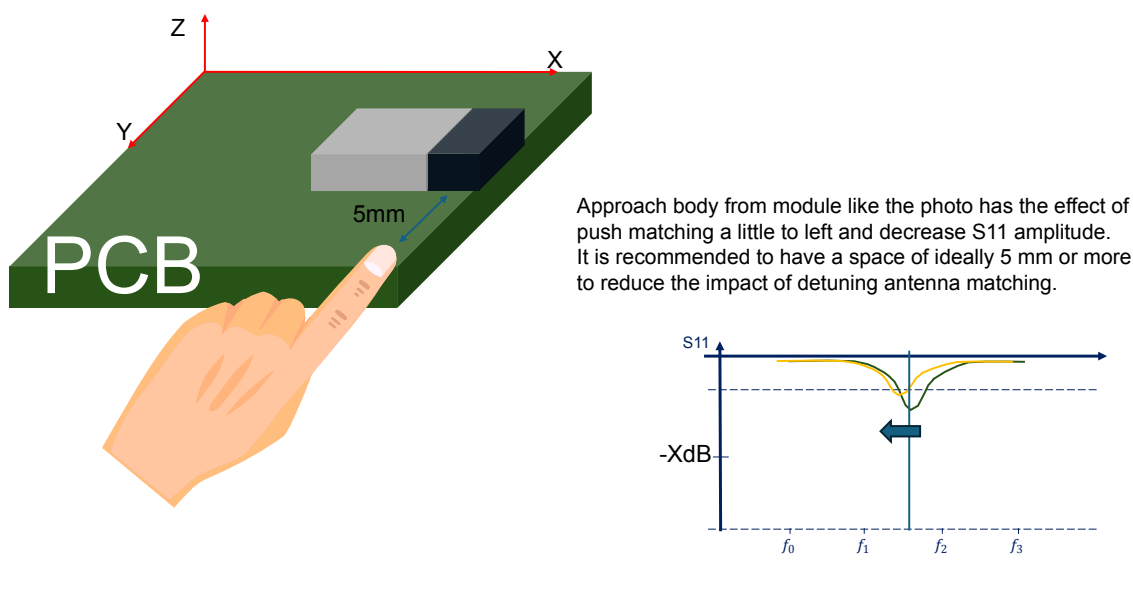
Body influence in Z down:

Figure 48. Effect of body approach toward module along Z- axis S11

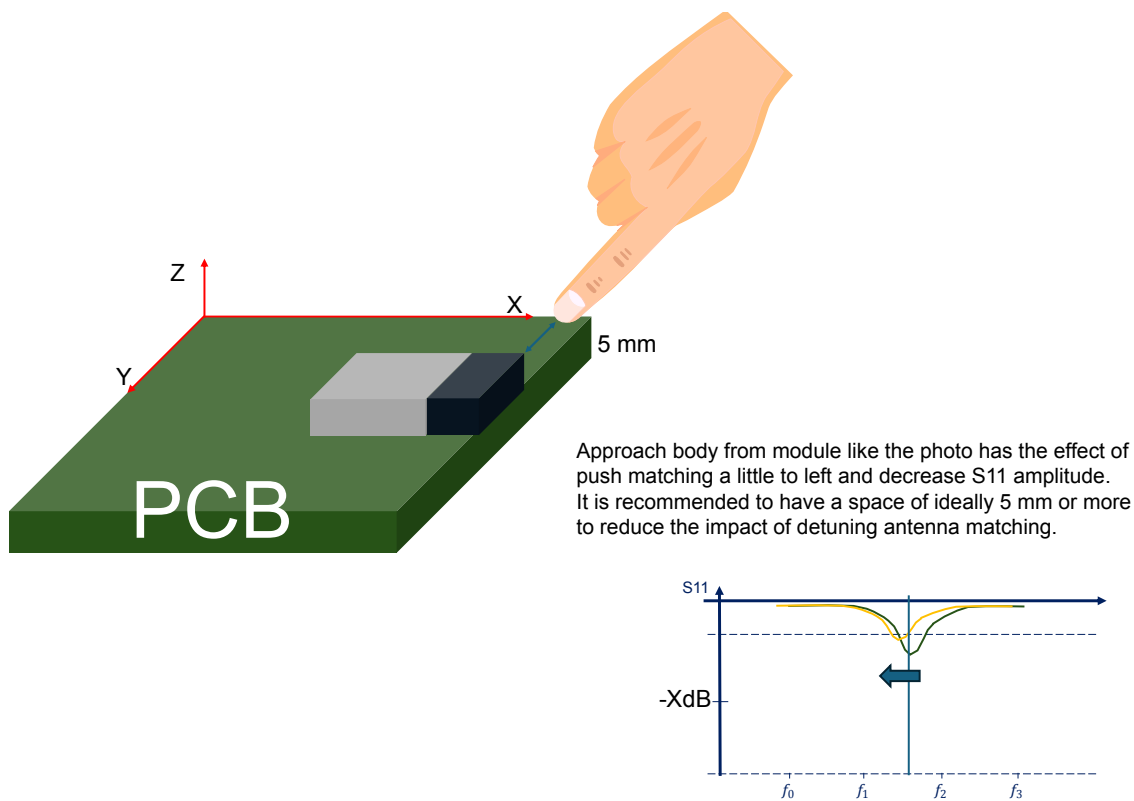


DT79068V1

Figure 49. Effect of body approach toward module along Y+ axis S11



DT79069V1

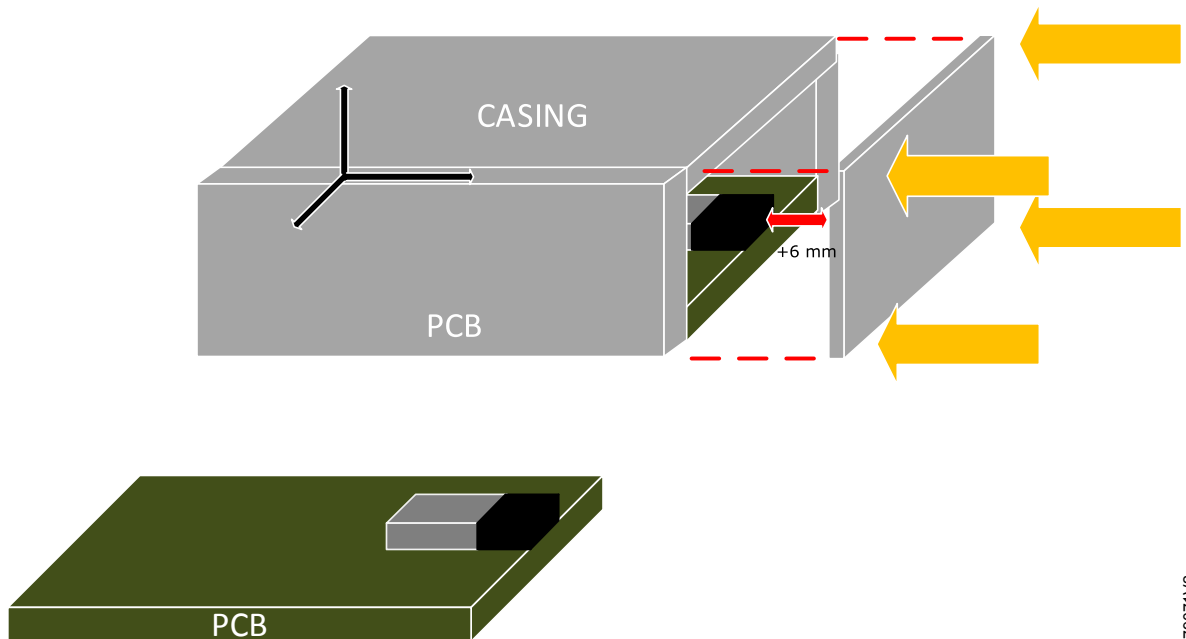
Figure 50. Effect of body approach toward module along Y- axis S11


DT79070V1

10 Plastic case

Normally, the products of the customer is in some sort of plastic or casing/housing. For this, some cautions should be taken in account:

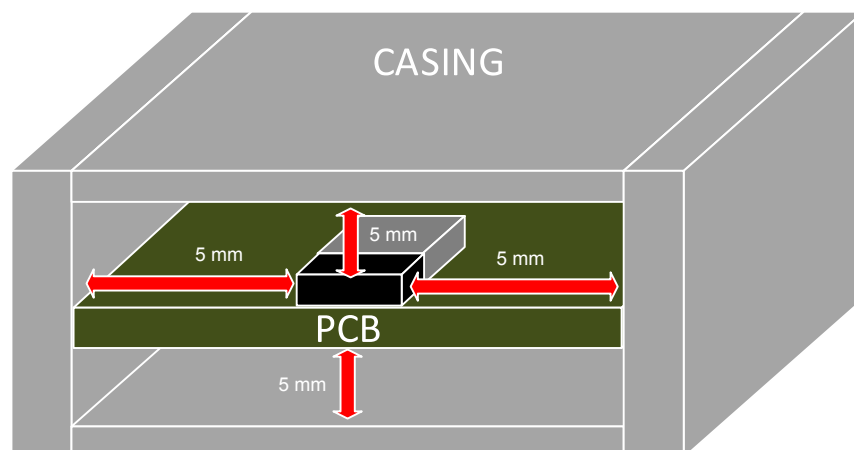
Figure 51. Plastic casing/housing and user normal board



DT79071V2

The casing should not be too close to the module. If possible, it is advised to maintain some space to avoid RF degradation. Here are some suggested spacing guidelines:

Figure 52. Plastic case suggested clearance dimensions



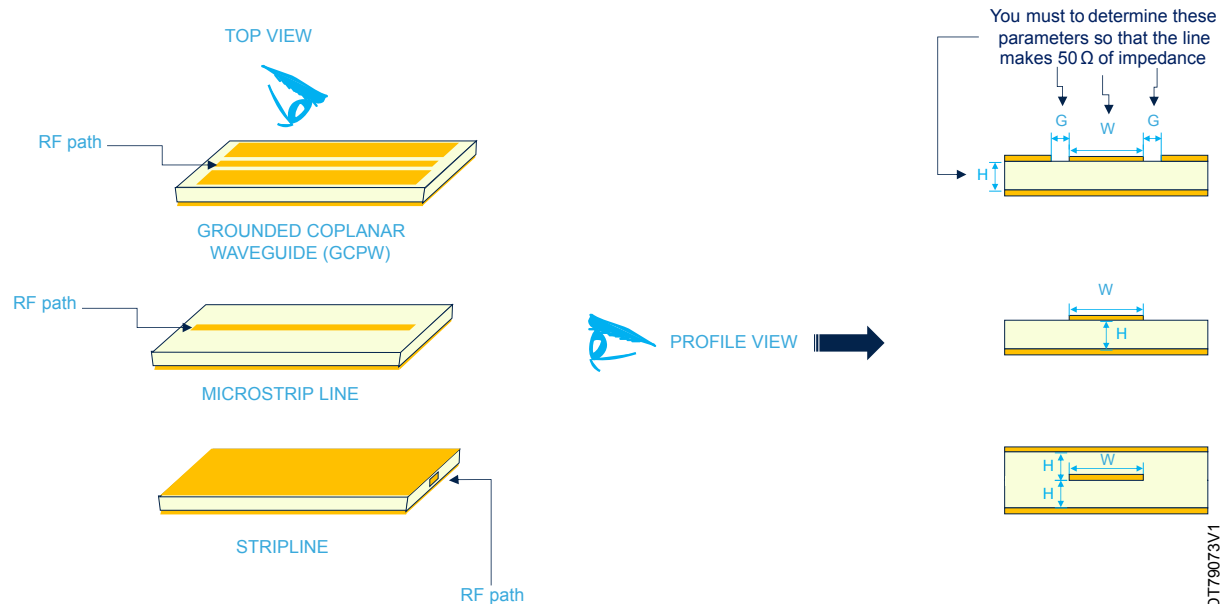
DT79072V2

Keep in mind that some materials are filled with metal particles, which can lead to even greater degradation. This is the case with some black ink used in black casings.

11 RF track

If the customer wishes to measure conducted power or use an external antenna, they need to be aware of the 50 Ω RF track. Different types of RF tracks can be used to allow 50 Ω :

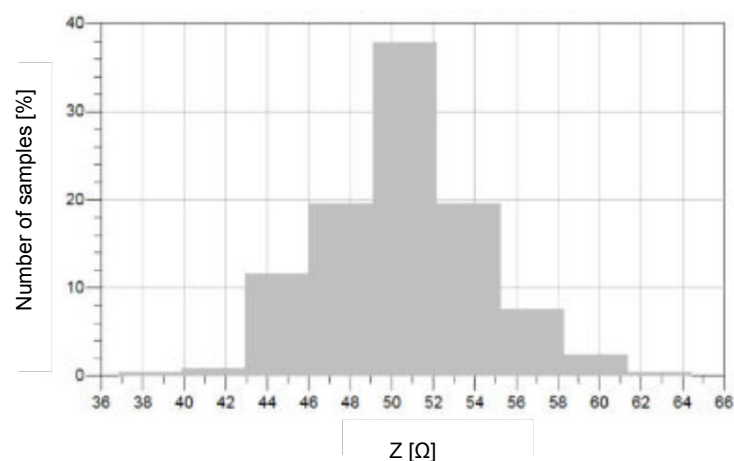
Figure 53. Different types of RF track



The most usual RF tracks are microstrip and coplanar since they are easier to project.

The PCB supplier can assist with achieving a controlled impedance of 50 Ω . Keep in mind that, due to PCB process variations, the parameters G, H, and W may vary slightly. These variations allow for achieving impedance distribution during manufacturing, as shown below:

Figure 54. Characteristic impedance

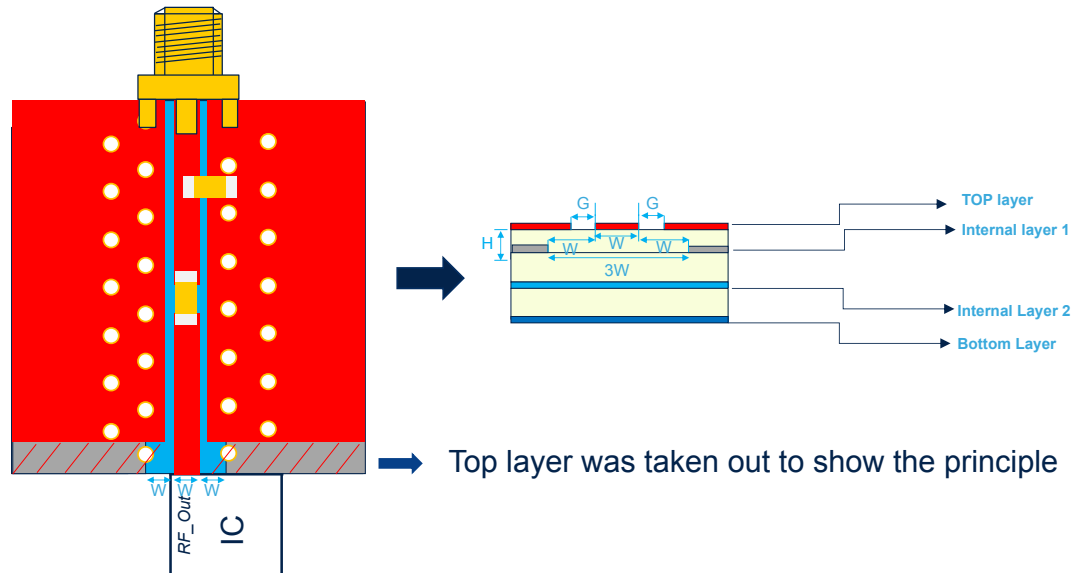


In some cases, a cutout may be needed if the PCB is too thin. For example, if microvias are used in PCB, this implies that the distance from the top layer to the midlayer is very thin, less than 0.120 mm, making it difficult to achieve 50 Ω in this configuration.

In this case, a metal cutout is most likely needed. Make a cut in the GND with a width of 3 W (where W is equal to the RF track width).

Depending on the design cutout could be needed in one or more layers). Speak with your PCB supplier or use RF simulation software's for example to know if the cutout is necessary.

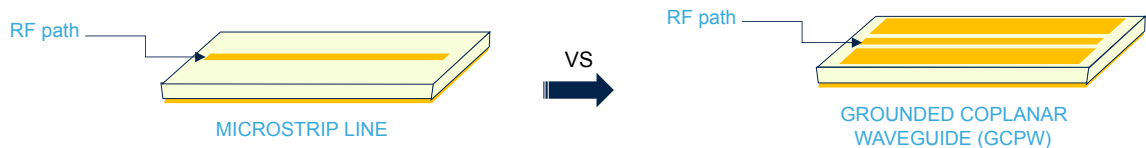
Figure 55. Cut-out



DT79075V1

The comparison of the main differences between microstrip lines and grounded coplanar waveguide (GCPW) tracks, is focusing on their popularity in PCB transmission lines:

Figure 56. Comparison microstrip line vs. grounded coplanar waveguide (GCPW)



Microstrip:

- Are less affected by variation of manufacturing process
- Have more radiation loss than GCPW
- Are more susceptible to spurious modes
- Are more dispersive and can limit bandwidth
- Easy to manufacture, no vias, just two layers
- Generally have lower loss than GCPW

GCPW:

- More robust in suppressing spurious modes
- More conductor loss but less radiation loss
- Less EM radiation, thus causing less interference
- More sensitive than microstrip to PCB fabrication processes
- Less coupling due to guard traces
- Circuit size is more compact
- Less dispersion than microstrip

Until 5 GHz the insertion loss is approximately the same for GCPW and microstrip line.

DT79076V1

12 Antenna maximum power radiation direction and others

In the ST anechoic chamber, various boards are tested in different configurations to provide customers with insights into board performance.

Figure 57. ST anechoic chamber

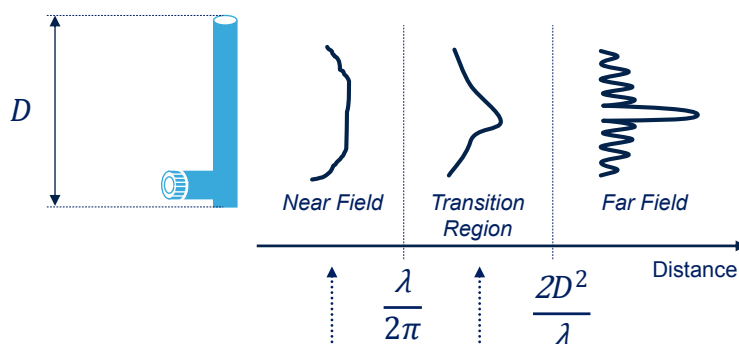


DT79077V1

The setup to pass our boards is aligned with certified labs and with certifications like FCC and RED. The module board transmits power in a distance of 3 m to the receiver (this region and beyond considered far field).

A brief memo about the far field region can be found below (classical formula found in literature):

Figure 58. Regions of electromagnetic fields (EM)–Near vs far field



If you place the Tx and Rx boards with antennas within these regions, you should expect strange behavior

DT79078V1

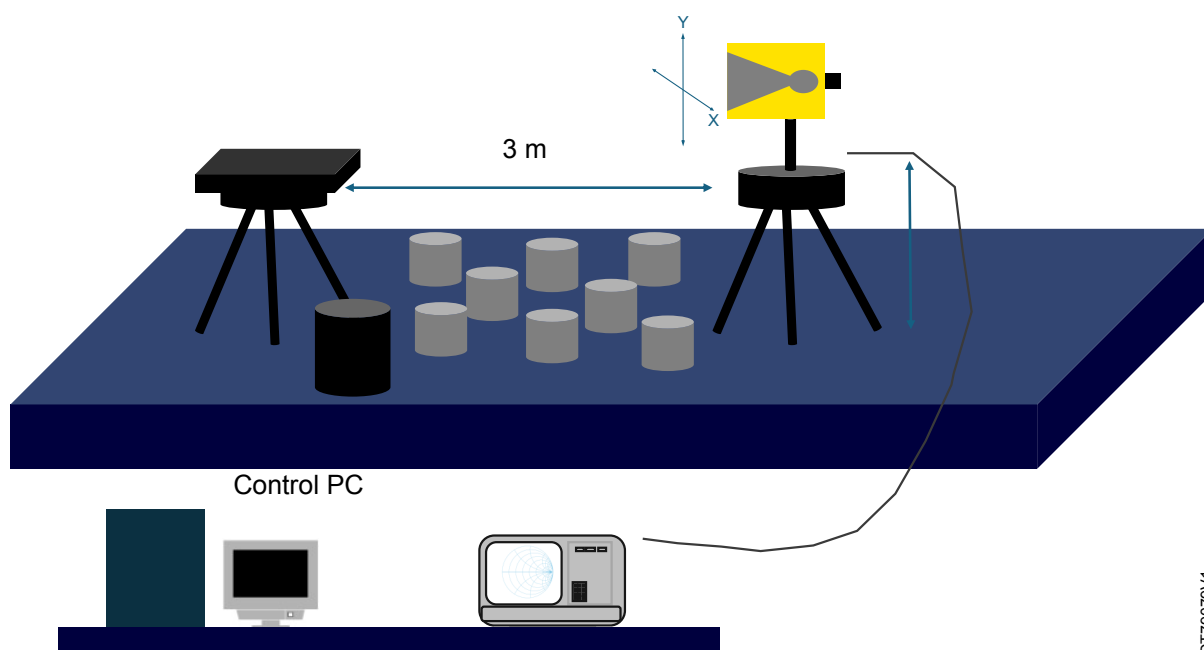
The electromagnetic (EM) field regions are detailed in the following table:

Table 1. Transition region for different antennas at 2.45 GHz

Antenna dimension	Transition zone
Half-wave dipole ($D = \lambda/2$)	~6.12 cm
Quarter-wavelength with GND plane ($D = \lambda/4$)	~1.53 cm

Antenna this configuration ID is done with receiver antenna in X polarization and Y polarization see figure below:

Figure 59. Anechoic chamber setup

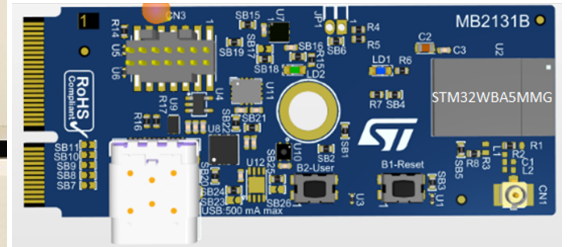
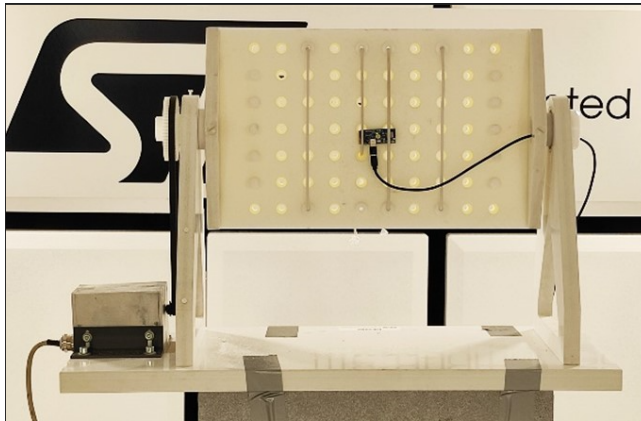


DT79079V1

Moreover, the module antenna board is placed on different axes and rotated to cover a 3D radiation pattern.

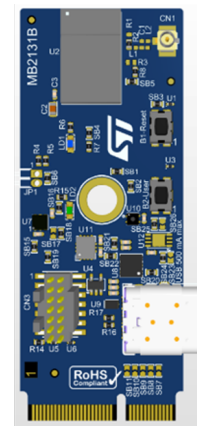
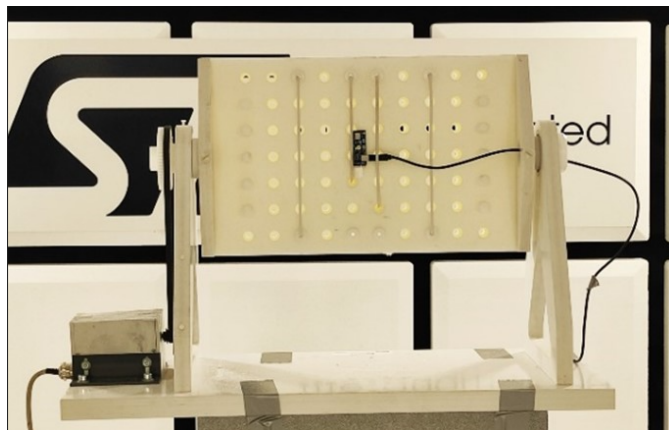
Figure 60. DUT test configurations

X board position



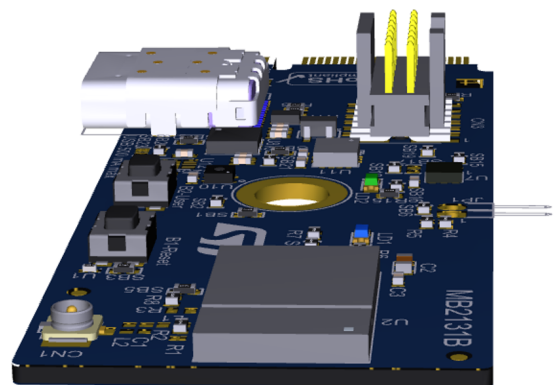
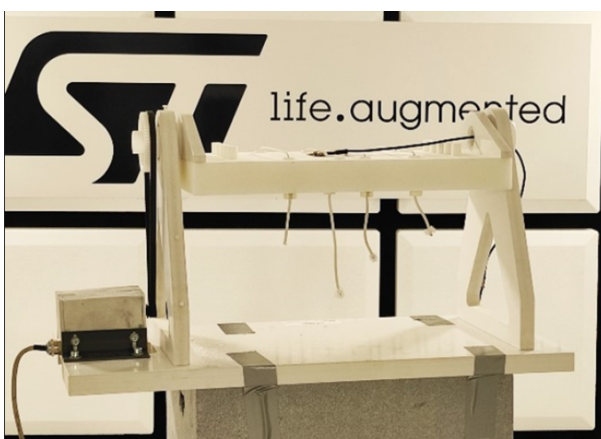
DT7565V1

Y board position



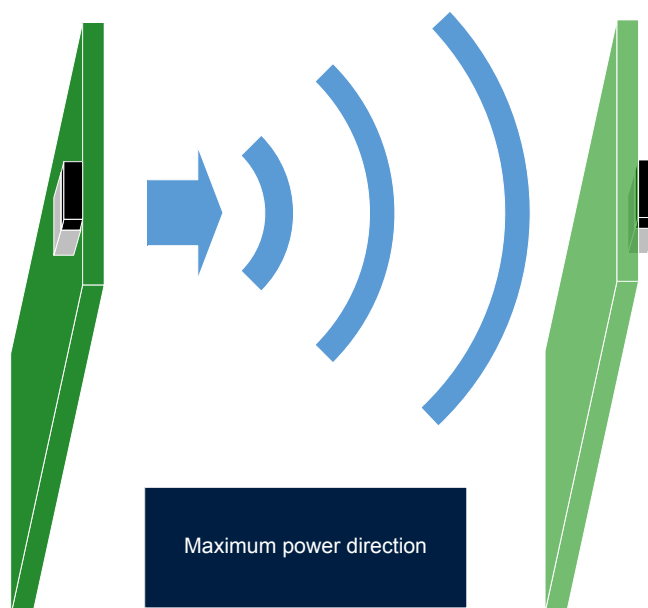
DT7565V1

Z board position



DT7565V1

This setup allows the identification of the optimal position of the module in space to deliver maximum power. This can help customers in applications requiring longer ranges. For example, the best position for the CEB board, a commercial product, is shown below:

Figure 61. Maximum radiation power module placement


DT79080V1

Note: *The radiation pattern may change if guidelines are not properly followed or if metal parts are used within the casing close to the module. Based on the application and required range, the product should be placed carefully.*

13 Conclusions

This application note provides guidelines for customers on module placement on a board and general topics related to module implementation, focusing on antenna performance. Many topics were covered, such as the basics of antennas, a general view of the module, clearance dimensions, and the impact of board dimensions on antenna performance. Additionally, fine-tuning of the antenna, the impact of casing, and the best module disposition for maximum power pattern were discussed.

Revision history

Table 2. Document revision history

Date	Revision	Changes
13-Jun-2025	1	Initial release.
12-Oct-2026	2	Updated title and Introduction on cover page. Updated Section 1: General information. Updated Figure 51. Plastic casing/housing and user normal board and Figure 52. Plastic case suggested clearance dimensions. Minor text changes.

Contents

1	General information	2
2	General considerations	3
3	Basics of antenna parameters	6
3.1	dBm and Watt relation	6
3.2	S11 parameters	7
3.3	Polarization	9
3.4	Antenna gain (G)	10
3.5	Antenna directivity (D)	10
3.6	Antenna efficiency (ϵ)	10
3.7	Antenna radiation pattern	11
4	Module internal antenna	15
5	Module placement	26
6	Layout mistakes to avoid	28
7	External antenna	31
8	Schematics	32
9	Antenna supplementary constraints	33
10	Plastic case	37
11	RF track	38
12	Antenna maximum power radiation direction and others	40
13	Conclusions	44
	Revision history	45
	List of tables	47
	List of figures	48

List of tables

Table 1.	Transition region for different antennas at 2.45 GHz	41
Table 2.	Document revision history	45

List of figures

Figure 1.	Module top and bottom view	3
Figure 2.	Module general view	3
Figure 3.	Top view showing the pins below	4
Figure 4.	Module block diagram	5
Figure 5.	Some basic correspondences between watt and dBm	6
Figure 6.	S-parameters view black box	7
Figure 7.	General view relation of S11 and reflected power	8
Figure 8.	Polarization coupling	9
Figure 9.	Directivity view isotropic antenna and directional antenna	10
Figure 10.	Radiation pattern plane view	11
Figure 11.	Radiation pattern planes view 2	11
Figure 12.	Radiation pattern antenna isotropic	12
Figure 13.	Radiation pattern antenna dipole	12
Figure 14.	Radiation pattern antenna Vivaldi	13
Figure 15.	Dbi and Dbd gain example for antenna Vivaldi	13
Figure 16.	Dbi and Dbd relation	14
Figure 17.	Module internal antenna and current path	15
Figure 18.	Relation between antenna clearance and S11	15
Figure 19.	Clearance for reference boards	16
Figure 20.	Connectivity expansion board (CEB)	16
Figure 21.	Option 2: fine-tuning	17
Figure 22.	Step 1 fine-tuning	18
Figure 23.	Step 2 fine-tuning	19
Figure 24.	Step 3 fine-tuning	20
Figure 25.	4 layer board global view tuning	21
Figure 26.	Top layer guidelines	21
Figure 27.	Midlayer 1	22
Figure 28.	Midlayer 1 guidelines	22
Figure 29.	Bottom layer	23
Figure 30.	Connection between antenna out and antenna in	24
Figure 31.	Antenna S11 measurement with probe connection	25
Figure 32.	Module best placement in board	26
Figure 33.	Module performance view with bigger board X axis (clearance axis)	26
Figure 34.	Module performance view with bigger boards Y axis	27
Figure 35.	Error 1: do not connect NC pads on the left	28
Figure 36.	Error 2: do not place GND in antenna clearance	28
Figure 37.	Error 3: do not place GND above antenna on the board	29
Figure 38.	Error 4: do not connect GND to NC pads on the right	29
Figure 39.	Error 4: avoid GND below on the right, if possible	30
Figure 40.	Module RF configurations	31
Figure 41.	Module designs with external antenna	31
Figure 42.	Antenna in and antenna out pin view	32
Figure 43.	CEB schematics module part	32
Figure 44.	Module antenna placement view	33
Figure 45.	Approach of external object toward module along main axes	33
Figure 46.	Effect of body approach toward module along X+ axis S11	34
Figure 47.	Effect of body approach toward module along Z+ axis S11	34
Figure 48.	Effect of body approach toward module along Z- axis S11	35
Figure 49.	Effect of body approach toward module along Y+ axis S11	35
Figure 50.	Effect of body approach toward module along Y- axis S11	36
Figure 51.	Plastic casing/housing and user normal board	37
Figure 52.	Plastic case suggested clearance dimensions	37
Figure 53.	Different types of RF track	38

Figure 54.	Characteristic impedance	38
Figure 55.	Cut-out	39
Figure 56.	Comparison microstrip line vs. grounded coplanar waveguide (GCPW)	39
Figure 57.	ST anechoic chamber	40
Figure 58.	Regions of electromagnetic fields (EM)—Near vs far field	41
Figure 59.	Anechoic chamber setup	41
Figure 60.	DUT test configurations	42
Figure 61.	Maximum radiation power module placement	43

IMPORTANT NOTICE – READ CAREFULLY

STMicroelectronics NV and its subsidiaries ("ST") reserve the right to make changes, corrections, enhancements, modifications, and improvements to ST products and/or to this document at any time without notice.

In the event of any conflict between the provisions of this document and the provisions of any contractual arrangement in force between the purchasers and ST, the provisions of such contractual arrangement shall prevail.

The purchasers should obtain the latest relevant information on ST products before placing orders. ST products are sold pursuant to ST's terms and conditions of sale in place at the time of order acknowledgment.

The purchasers are solely responsible for the choice, selection, and use of ST products and ST assumes no liability for application assistance or the design of the purchasers' products.

No license, express or implied, to any intellectual property right is granted by ST herein.

Resale of ST products with provisions different from the information set forth herein shall void any warranty granted by ST for such product.

If the purchasers identify an ST product that meets their functional and performance requirements but that is not designated for the purchasers' market segment, the purchasers shall contact ST for more information.

ST and the ST logo are trademarks of ST. For additional information about ST trademarks, refer to www.st.com/trademarks. All other product or service names are the property of their respective owners.

Information in this document supersedes and replaces information previously supplied in any prior versions of this document.

© 2026 STMicroelectronics – All rights reserved