

# Application Note

## Designing Printed Antennas for Bluetooth® Low Energy

AN-B-027

### Abstract

*Guidelines for designing printed antennas for 2.4 GHz Bluetooth® low energy applications on single-layer and multi-layer PCB stack-ups.*

## Designing Printed Antennas for Bluetooth® Low Energy

### Contents

<b>Abstract .....</b>	<b>1</b>
<b>Contents .....</b>	<b>2</b>
<b>Figures.....</b>	<b>3</b>
<b>1 Terms and Definitions.....</b>	<b>4</b>
<b>2 References .....</b>	<b>4</b>
<b>3 Introduction.....</b>	<b>4</b>
<b>4 RF Layout Design.....</b>	<b>5</b>
4.1 Radio IC .....	5
4.2 RF Transmission Lines, Filters, and Matching Networks.....	6
4.3 Antennas, General Considerations .....	6
<b>5 Antennas on Multi-Layer Substrates.....</b>	<b>7</b>
5.1 Full Size Printed IFA, 1 mm Substrate .....	8
5.1.1 Matching Network .....	9
5.1.2 Measured Return Loss .....	9
5.1.3 Measured Radiation Pattern .....	10
5.2 Reduced Size Printed IFA, 1 mm Substrate, Variant 1 .....	11
5.2.1 Matching Network .....	12
5.2.2 Measured Return Loss .....	12
5.2.3 Measured Radiation Pattern .....	13
5.3 Reduced Size Printed IFA, 1 mm Substrate, Variant 2.....	14
5.3.1 Matching Network .....	15
5.3.2 Measured Return Loss .....	15
5.3.3 Measured Radiation Pattern .....	16
5.4 ZOR Antenna, 1.6 mm Substrate.....	17
5.4.1 Matching Network .....	18
5.4.2 Measured Return Loss .....	18
5.4.3 Measured Radiation Pattern .....	19
5.5 ZOR Antenna, 1.0 mm Substrate.....	20
5.5.1 Matching Network .....	21
5.5.2 Measured Return Loss .....	21
5.5.3 Measured Radiation Pattern .....	22
<b>6 Antennas on Single Layer Substrates .....</b>	<b>23</b>
6.1 Carbon Layers.....	23
6.2 Single Layer Printed IFA, 1 mm Substrate.....	24
6.2.1 Matching Network .....	25
6.2.2 Measured Return Loss .....	25
6.2.3 Measured Radiation Pattern .....	26
<b>7 Conclusions .....</b>	<b>27</b>
<b>8 Revision History .....</b>	<b>28</b>

## Designing Printed Antennas for Bluetooth® Low Energy

### Figures

Figure 1: DA1458x_CB_PXI_WLCSP Board Layout Snapshot .....	5
Figure 2: Placement of Filters and Matching Networks.....	6
Figure 3: Full Size IFA with Dimensions.....	8
Figure 4: Matching Components, Full Size IFA.....	9
Figure 5: Measured S11, Full Size IFA .....	9
Figure 6: Radiation Pattern, PCB Horizontal, Full Size IFA .....	10
Figure 7: Radiation Pattern, PCB Vertical, Full Size IFA .....	10
Figure 8: Reduced Size IFA, Variant 1 .....	11
Figure 9: Matching Components, Reduced Size IFA, Variant 1.....	12
Figure 10: Measured S11, Reduced Size IFA, Variant 1 .....	12
Figure 11: Radiation Pattern, PCB Horizontal, Reduced Size IFA, Variant 1 .....	13
Figure 12: Radiation Pattern, PCB Vertical, Reduced Size IFA, Variant 1 .....	13
Figure 13: Reduced Size IFA, Variant 2.....	14
Figure 14: Matching Components, Reduced Size IFA, Variant 2 .....	15
Figure 15: Measured S11, Reduced Size IFA, Variant 2 .....	15
Figure 16: Radiation Pattern, PCB Horizontal, Reduced Size IFA, Variant 2 .....	16
Figure 17: Radiation Pattern, PCB Vertical, Reduced Size IFA, Variant 2 .....	16
Figure 18: ZOR Antenna Preview .....	17
Figure 19: Matching Components, ZOR Antenna, 1.6 mm Substrate .....	18
Figure 20: Measured S11, ZOR Aantenna, 1.6 mm Substrate .....	18
Figure 21: Radiation Pattern, DUT Horizontal, ZOR Antenna, 1.6 mm Substrate .....	19
Figure 22: Radiation Pattern, DUT Vertical, ZOR Antenna, 1.6 mm Substrate .....	19
Figure 23: ZOR Antenna Preview .....	20
Figure 24: Matching Components, ZOR Antenna, 1.0 mm Substrate .....	21
Figure 25: Measured S11, ZOR Antenna, 1.0 mm Substrate .....	21
Figure 26: Radiation Pattern, DUT Horizontal, ZOR Antenna, 1.0 mm Substrate .....	22
Figure 27: Radiation Pattern, DUT Vertical, ZOR Antenna, 1.0 mm Substrate .....	22
Figure 28: Single Layer IFA.....	24
Figure 29: Matching Components, Single Layer IFA .....	25
Figure 30: Measured S11, Single Layer IFA .....	25
Figure 31: Radiation Pattern, PCB Horizontal, Single Layer IFA .....	26
Figure 32: Radiation Pattern, PCB Vertical, Single Layer IFA .....	26

---

## Designing Printed Antennas for Bluetooth® Low Energy

---

### 1 Terms and Definitions

balun	balanced/unbalanced (transformer)
BLE	Bluetooth low energy
DNP	Do Not Populate
GND	Ground (plane)
IC	Integrated Circuit
IFA	Inverted-F Antenna
PCB	Printed Circuit Board
XTAL	(quartz) crystal
ZOR	Zeroth-Order Resonance (antenna)

### 2 References

N/A

### 3 Introduction

This application note contains guidelines for implementing printed PCB antennas for Bluetooth® low energy applications operating in the 2.4 GHz frequency band.

Basic rules are given on what to avoid when starting with an antenna design.

For multi-layer PCBs several antenna options are given, depending on the available PCB space. There is a trade-off between antenna size and radiation performance. Full size antennas are more broadband and easier to match since the impedance is closer to 50  $\Omega$ .

For single-layer PCBs the main challenge is providing a proper ground (GND) plane for the antenna. When a lower quality substrate material is used to reduce cost, this will also have an impact on the radiation efficiency of the antenna.

## Designing Printed Antennas for Bluetooth® Low Energy

### 4 RF Layout Design

Guidelines to consider for general RF layout work.

#### 4.1 Radio IC

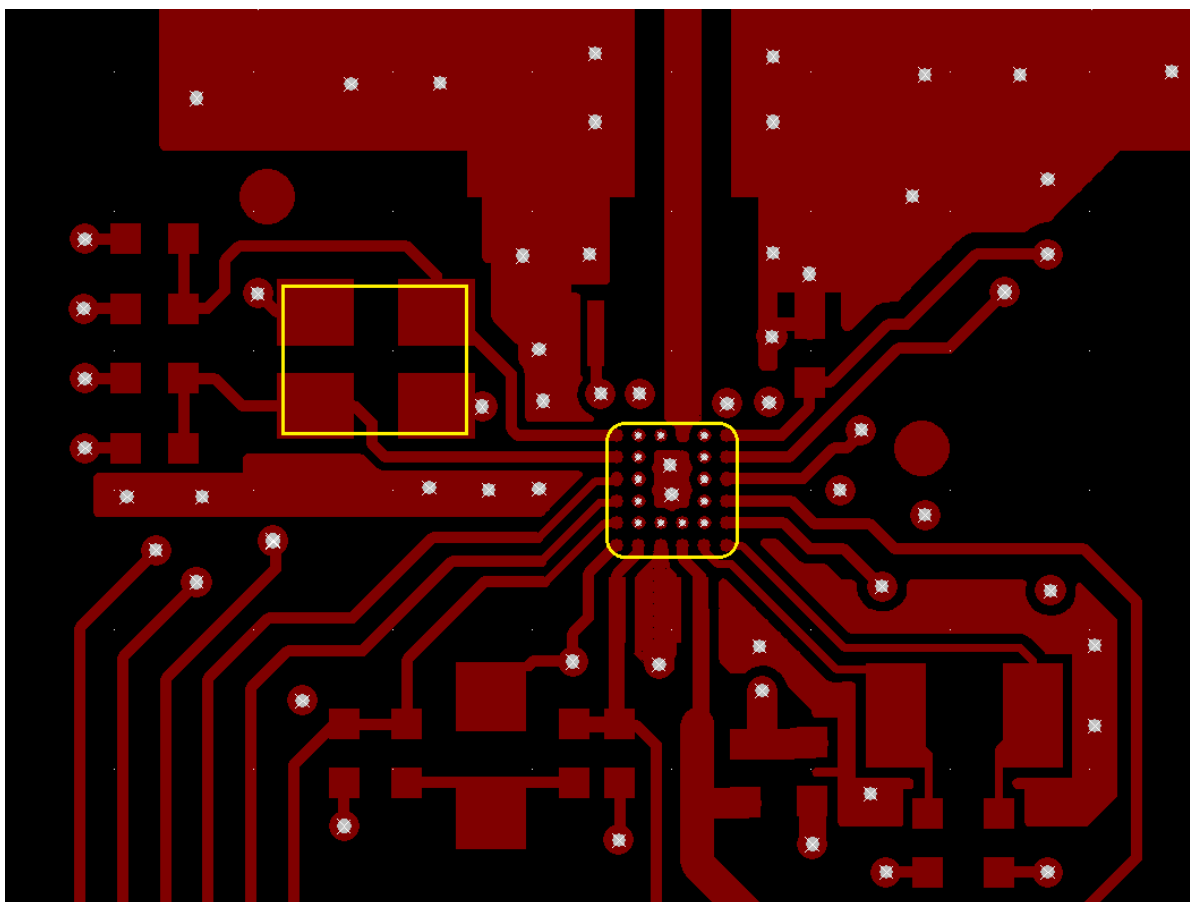
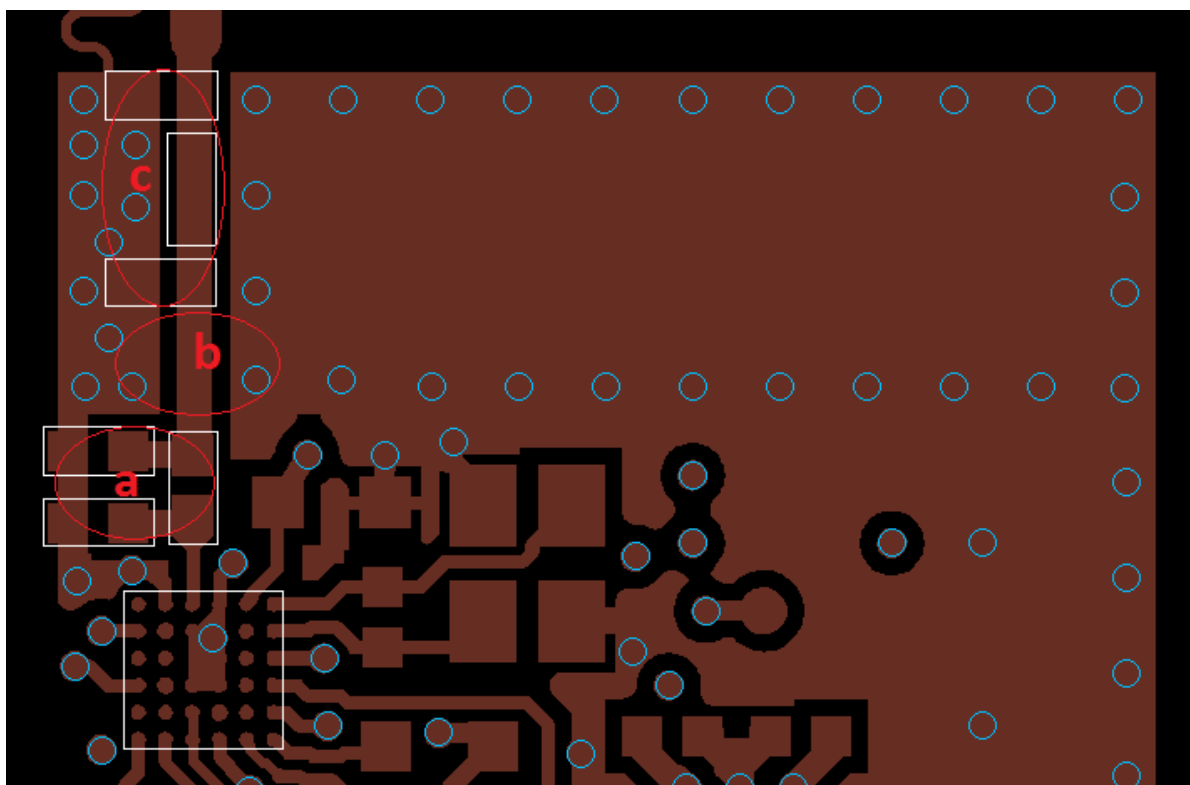


Figure 1: DA1458x\_CB\_PXI\_WLCSP Board Layout Snapshot

- Active components operating at high frequency should have as compact a layout as possible. This will prevent cross-coupling between lines and parasitics, which will have a negative effect on the operating parameters.
- Always provide a solid grounding to the radio IC. Use as many vias as possible to create a solid GND under the IC itself and connect it to inner and bottom GND layers.
- Place the XTAL used as reference for the RF carrier (16 MHz for the DA1458x) as close as possible to the IC. This minimizes any additional capacitive load on the input pins and reduces the chance of crosstalk and interference with other signals on the board.

## Designing Printed Antennas for Bluetooth® Low Energy

### 4.2 RF Transmission Lines, Filters, and Matching Networks



**Figure 2: Placement of Filters and Matching Networks**

- Place radio matching components and any RF filters as close as possible to the radio IC.
  - Make sure the component's GNDs are connected to the same GND plane (left side of the transmission line above).
- Minimize the transmission line length between radio IC and antenna.
  - The characteristic impedance of the transmission line should match the required radio impedance ( $50\ \Omega$  for the DA1458x).
  - Most current PCB layout tools have the possibility to calculate the characteristic impedance of the transmission line based on the stack-up and PCB material.
- Place antenna matching components as close as possible to the feeding point of the antenna.
  - Make sure the component's GNDs are connected to the same GND plane (left side of the transmission line above).

### 4.3 Antennas, General Considerations

Consider the following remarks before starting with an antenna design:

- Do not place metal layers below the antenna itself. The antenna footprint must be kept free of metal.
- Do not place metal screws, radiators, piezo buzzers, batteries, etc. in the proximity of the antenna.
  - As a rule of thumb, at least 5 mm should be allowed around the antenna footprint, both horizontally on the PCB and vertically around the PCB footprint.
- Do not use metal enclosures for products with antennas. This will prevent the antenna from radiating and performing as intended.

---

## Designing Printed Antennas for Bluetooth® Low Energy

---

### 5 Antennas on Multi-Layer Substrates

On a multi-layer PCB stack-up it is relatively easy to construct a good reference GND plane for the antenna. This is especially important because the most common antenna choice is the Inverted-F Antenna (IFA).

It is very important to understand that IFAs operate in conjunction with a GND plane. There is a minimum required size of the GND plane for efficient antenna operation. This parameter has been specifically indicated for all designs below.

Depending on the available space on the PCB one can choose between a full size IFA and a folded/shrunk version.

The full size IFA has more broadband frequency characteristics and better radiation efficiency. The folded versions can reduce the space requirements by half and still achieve 100 MHz to 150 MHz bandwidth, which is enough to cover the Bluetooth low energy 2.4 GHz band (at -10 dB return loss).

The achievable radiation efficiency is between -1.5 dB and -3 dB. Efficiency is affected by the PCB material, therefore it is recommended that FR4 (low loss) substrate material is used.

The antennas in the following sections were designed and measured using:

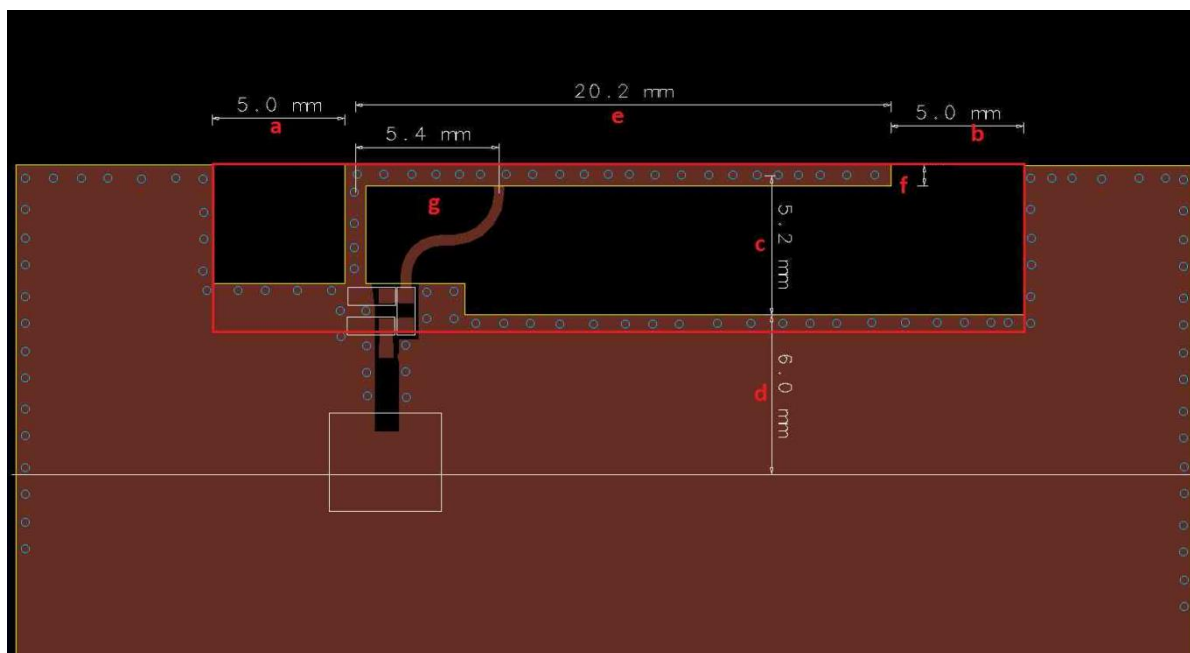
- FR4 PCB substrate material
- 2 or more metal (copper) layers

The dimensions given below are based on laboratory measurements and are optimised for the given stack-up and board size.

Variations in board thickness or substrate material will have an effect on the antenna matching. Also, when the antenna is in close proximity of a plastic enclosure the optimum dimensions and matching components may be different.

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.1 Full Size Printed IFA, 1 mm Substrate



**Figure 3: Full Size IFA with Dimensions**

The red outline indicates the antenna footprint, i.e. required allocation of PCB space. The antenna footprint is available on request in DXF format.

Legend (see [Figure 3](#)):

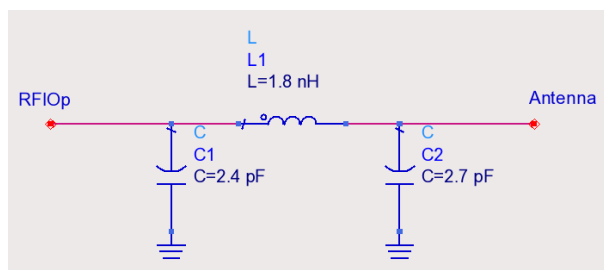
- a. Clearance between antenna arm and GND plane left.
- b. Clearance between antenna arm and GND plane right.
  - i. For narrow PCBs (a) and (b) will coincide with board edges.
  - ii. The two GND pieces left and right of the antenna are NOT required for correct antenna operation.
- c. Clearance between antenna arm and GND plane below.
- d. Minimum GND plane size required for correct operation of the antenna.
- e. Antenna width.
  - i. The antenna is implemented on top and bottom layers and stitched together using vias.
  - ii. The feeding line (from indicated matching components) is implemented on top layer only.
- f. Antenna trace width (0.6 mm).
- g. Feed point position.



## Designing Printed Antennas for Bluetooth® Low Energy

### 5.1.1 Matching Network

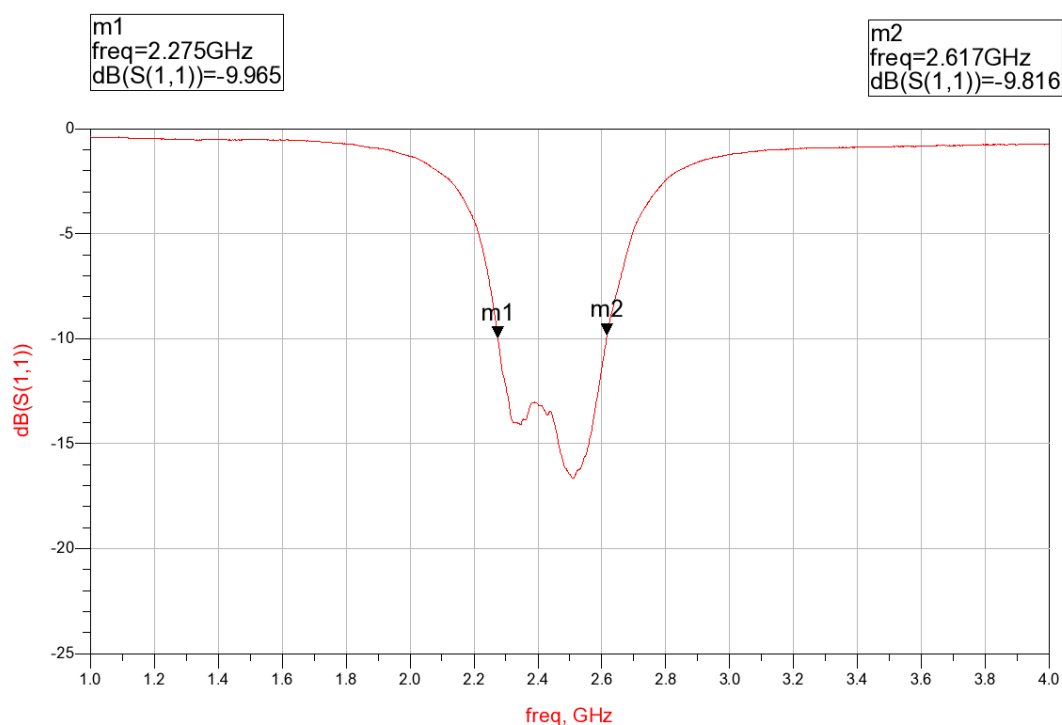
The matching is subject to change depending on substrate type or thickness and enclosure material (type and proximity to PCB).



**Figure 4: Matching Components, Full Size IFA**

- C1: 2.4 pF, 0402, GRM15 series, Murata
- L1: 1.8 nH, 0402, LQP series, Murata
- C2: 2.7 pF, 0402, GRM15 series, Murata
- In addition to the matching network, a 3.3 nH or 3.9 nH coil (depending on the DA1458x package) is required close to the RFIO pin.

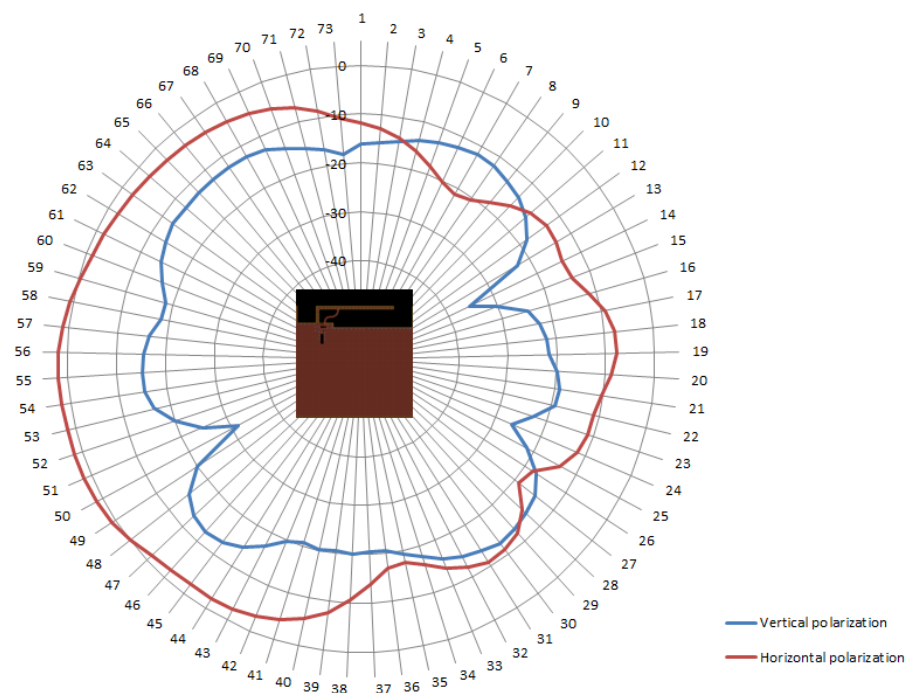
### 5.1.2 Measured Return Loss



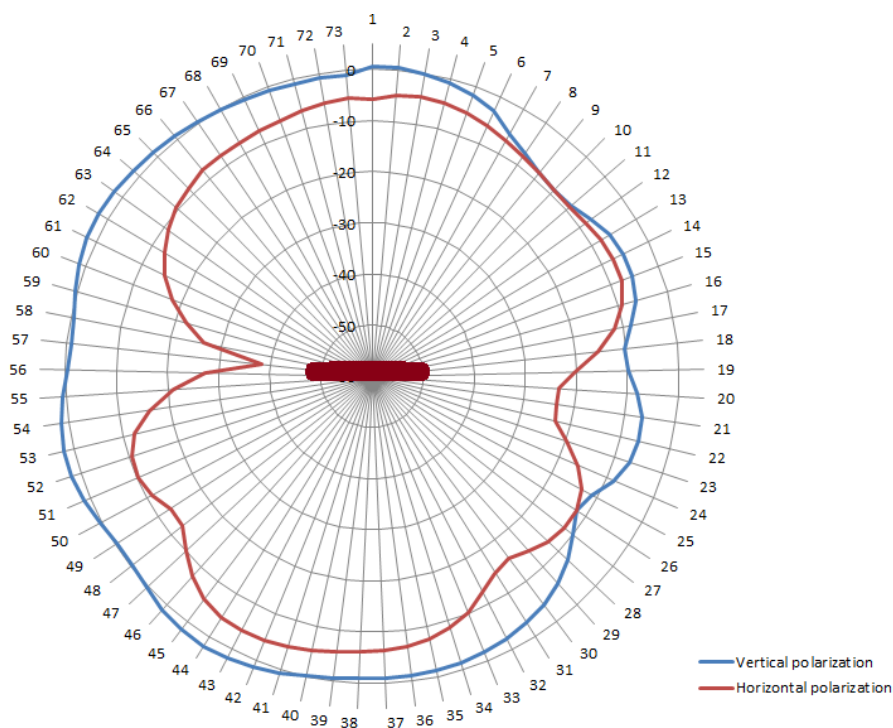
**Figure 5: Measured S11, Full Size IFA**

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.1.3 Measured Radiation Pattern



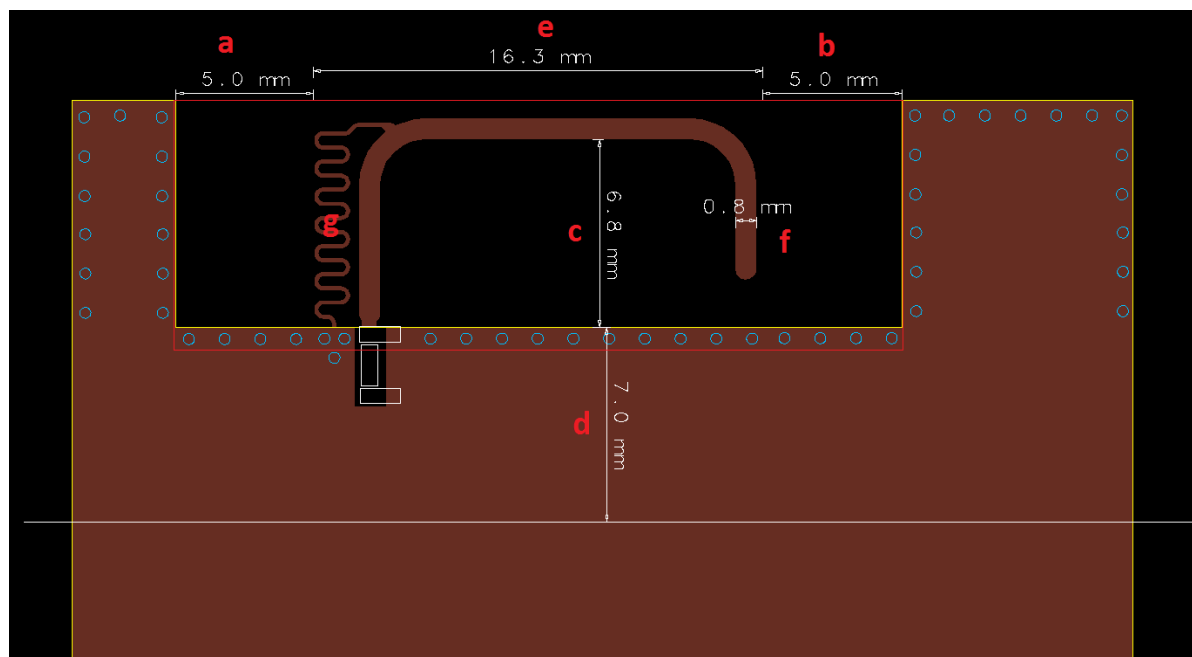
**Figure 6: Radiation Pattern, PCB Horizontal, Full Size IFA**



**Figure 7: Radiation Pattern, PCB Vertical, Full Size IFA**

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.2 Reduced Size Printed IFA, 1 mm Substrate, Variant 1



**Figure 8: Reduced Size IFA, Variant 1**

The red outline indicates the antenna footprint, i.e. required allocation of PCB space. The antenna is implemented on the TOP layer only. The antenna footprint is available on request in DXF format.

Legend (see [Figure 4](#)):

- a. Clearance between antenna and GND plane left.
- b. Clearance between antenna and GND plane right.
  - i. For narrow PCBs (a) and (b) will coincide with board edges.
  - ii. The two GND pieces left and right of the antenna are NOT required for correct antenna operation.
- c. Clearance between antenna arm and GND plane below.
- d. Minimum GND plane size required for correct operation of the antenna.
- e. Antenna width.
- f. Antenna main arm.
  - i. Width: 0.8 mm (30 mil).
  - ii. Length: 24.4 mm (961 mil).
- g. Antenna meander arm.
  - i. Implemented on the TOP layer only.
  - ii. Width: 0.153 mm (6 mil).
  - iii. Length: 21.22 mm (835.5 mil).

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.2.1 Matching Network

The matching is subject to change depending on substrate type or thickness and enclosure material (type and proximity to PCB).

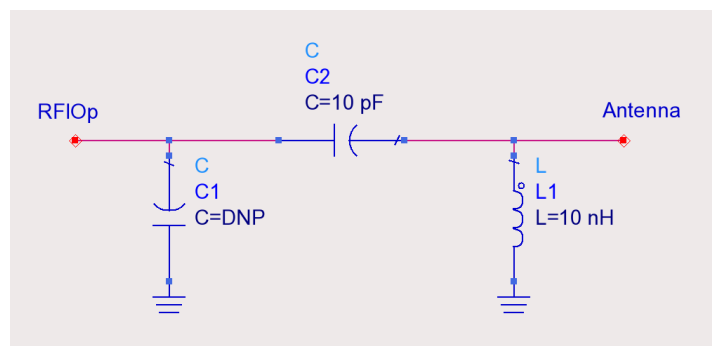


Figure 9: Matching Components, Reduced Size IFA, Variant 1

- C1: DNP
- C2: 10 pF, 0402, GRM15 series, Murata
- L1: 10 nH, 0402, LQP series, Murata
- In addition to the matching network, a 3.3 nH or 3.9 nH coil (depending on the DA1458x package) is required close to the RFIO pin.

### 5.2.2 Measured Return Loss

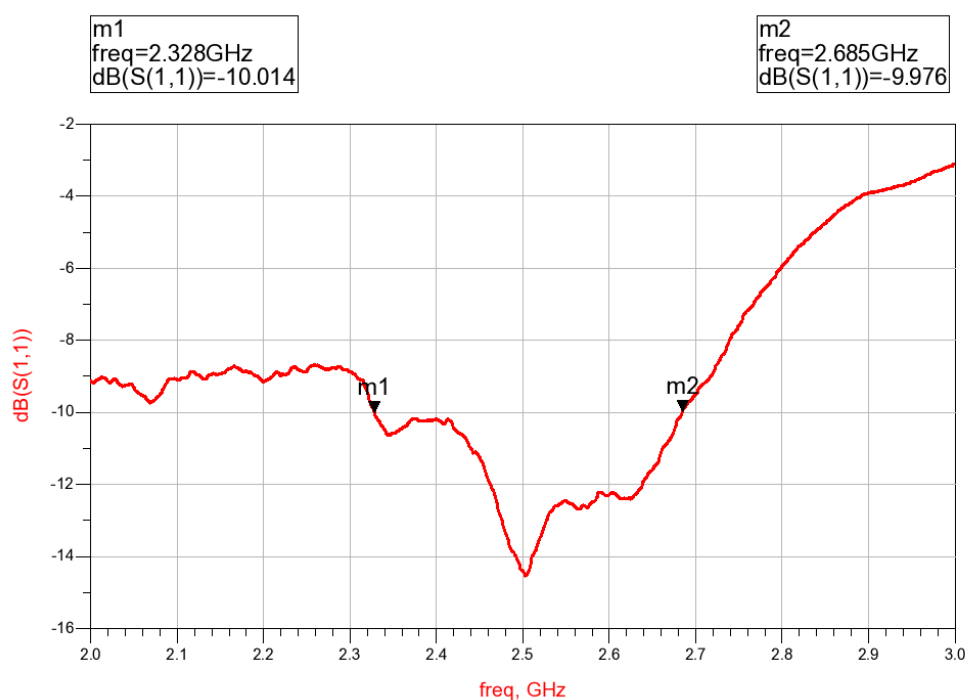


Figure 10: Measured S11, Reduced Size IFA, Variant 1

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.2.3 Measured Radiation Pattern

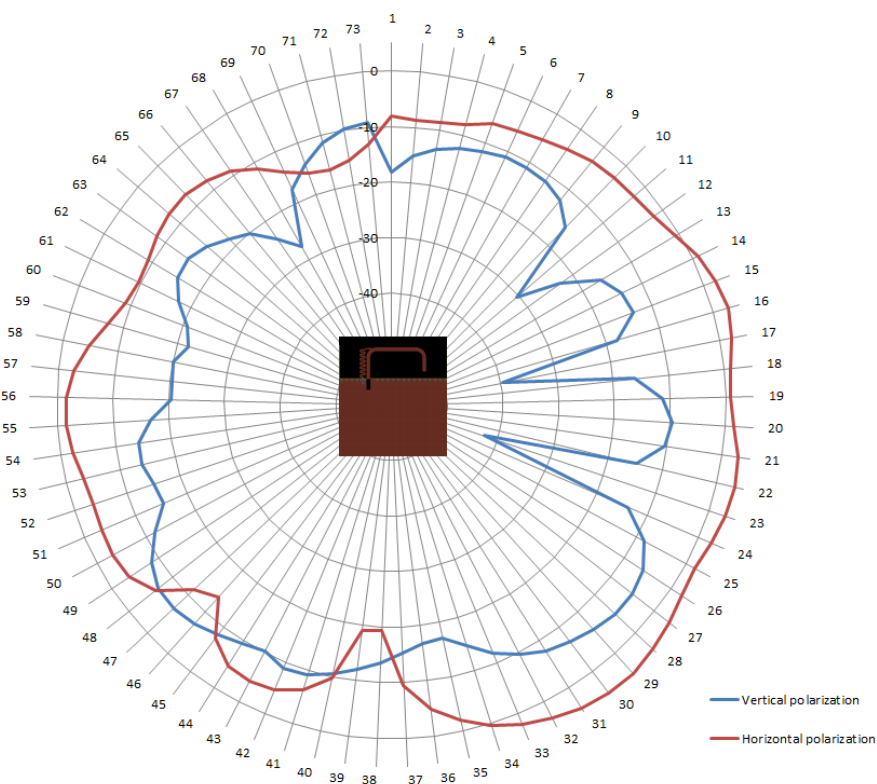


Figure 11: Radiation Pattern, PCB Horizontal, Reduced Size IFA, Variant 1

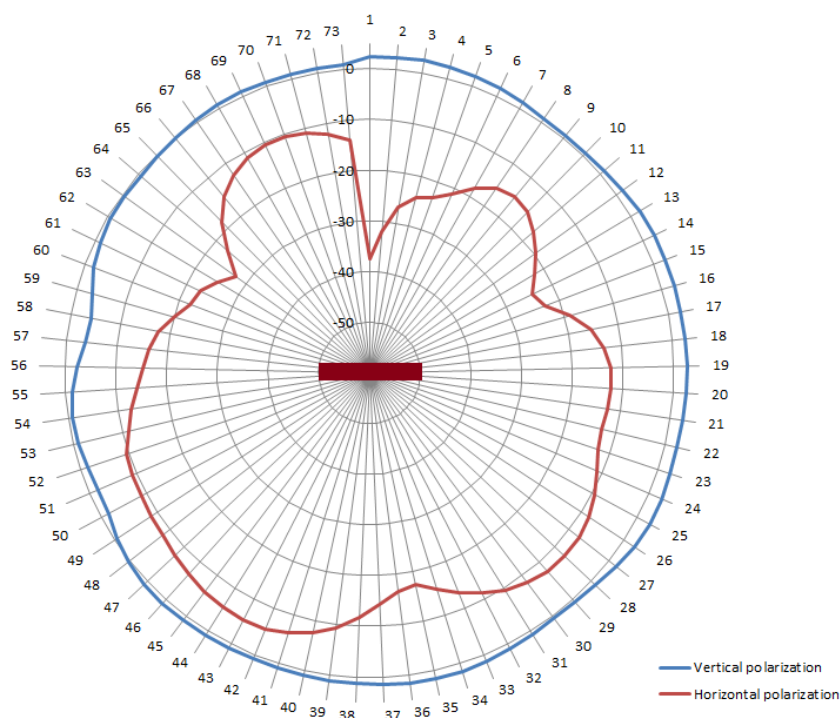
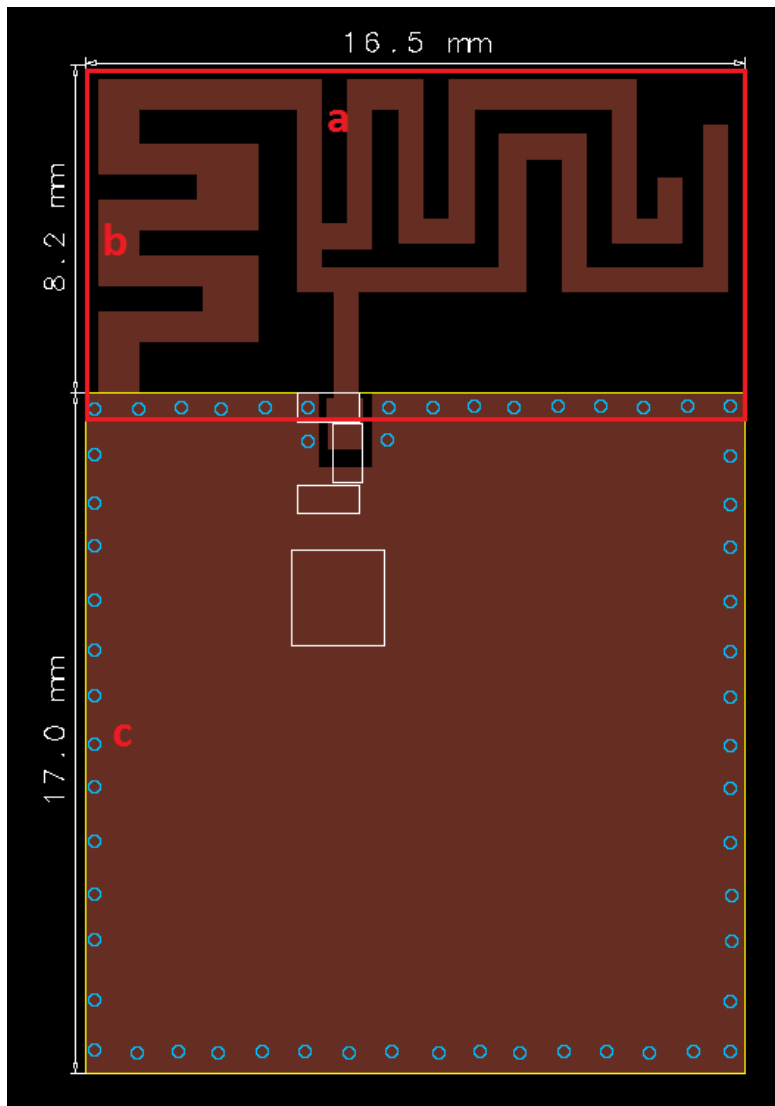


Figure 12: Radiation Pattern, PCB Vertical, Reduced Size IFA, Variant 1

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.3 Reduced Size IFA, 1 mm Substrate, Variant 2



**Figure 13: Reduced Size IFA, Variant 2**

The red outline indicates the antenna footprint, i.e. required allocation of PCB space. The antenna is implemented on the TOP layer only. The antenna footprint is available on request in DXF format.

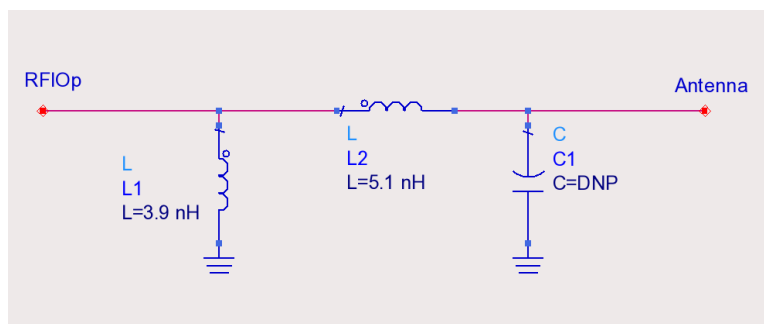
Legend (see [Figure 13](#)):

- a. Antenna footprint width.
- b. Antenna footprint height.
- c. Minimum GND plane size required for correct operation of the antenna.

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.3.1 Matching Network

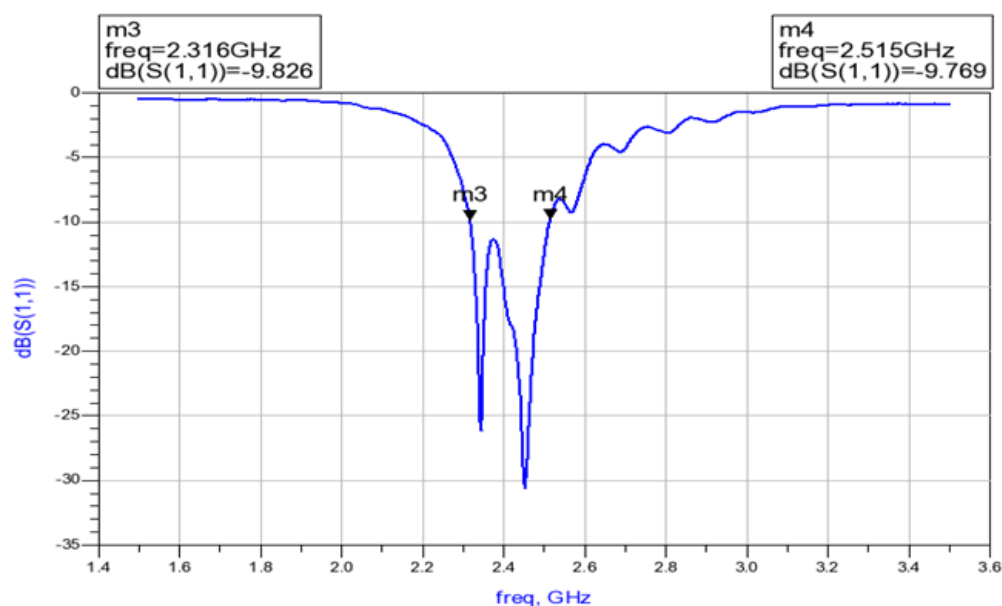
The matching is subject to change depending on substrate type or thickness and enclosure material (type and proximity to PCB).



**Figure 14: Matching Components, Reduced Size IFA, Variant 2**

- L1: 3.9 nH, 0402, LQP series, Murata
- L2: 5.1 nH, 0402, LQP series, Murata
- C1, DNP
- In addition to the matching network, a 3.3 nH or 3.9 nH coil (depending on the DA1458x package) is required close to the RFIO pin.

### 5.3.2 Measured Return Loss



**Figure 15: Measured S11, Reduced Size IFA, Variant 2**

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.3.3 Measured Radiation Pattern

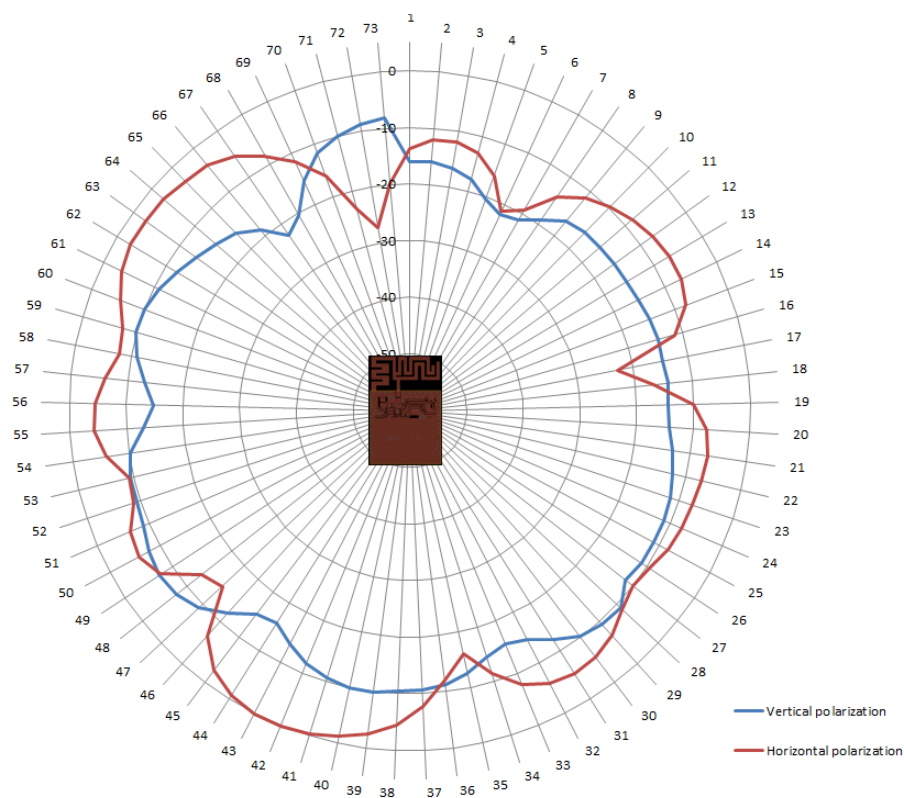


Figure 16: Radiation Pattern, PCB Horizontal, Reduced Size IFA, Variant 2

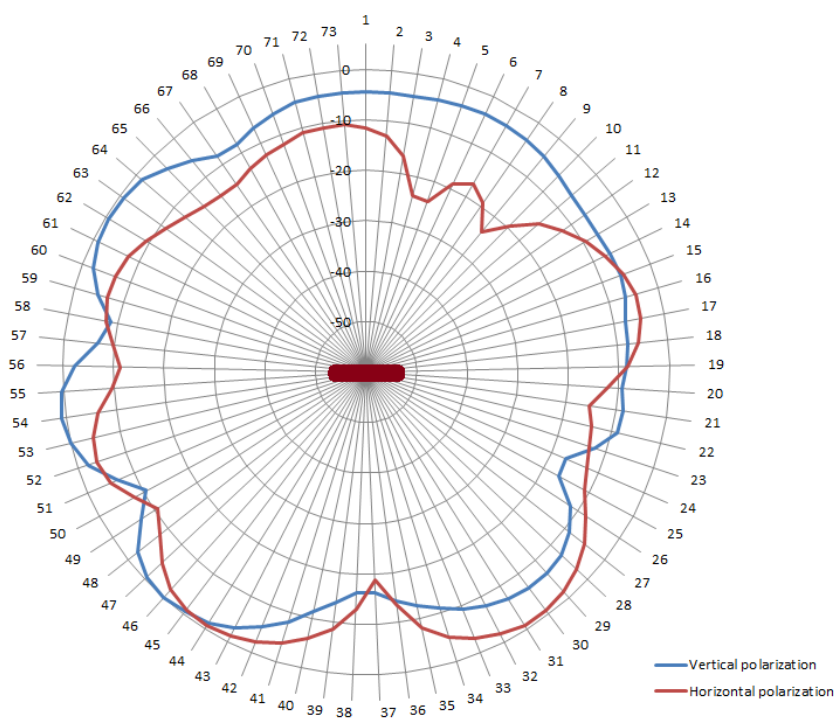


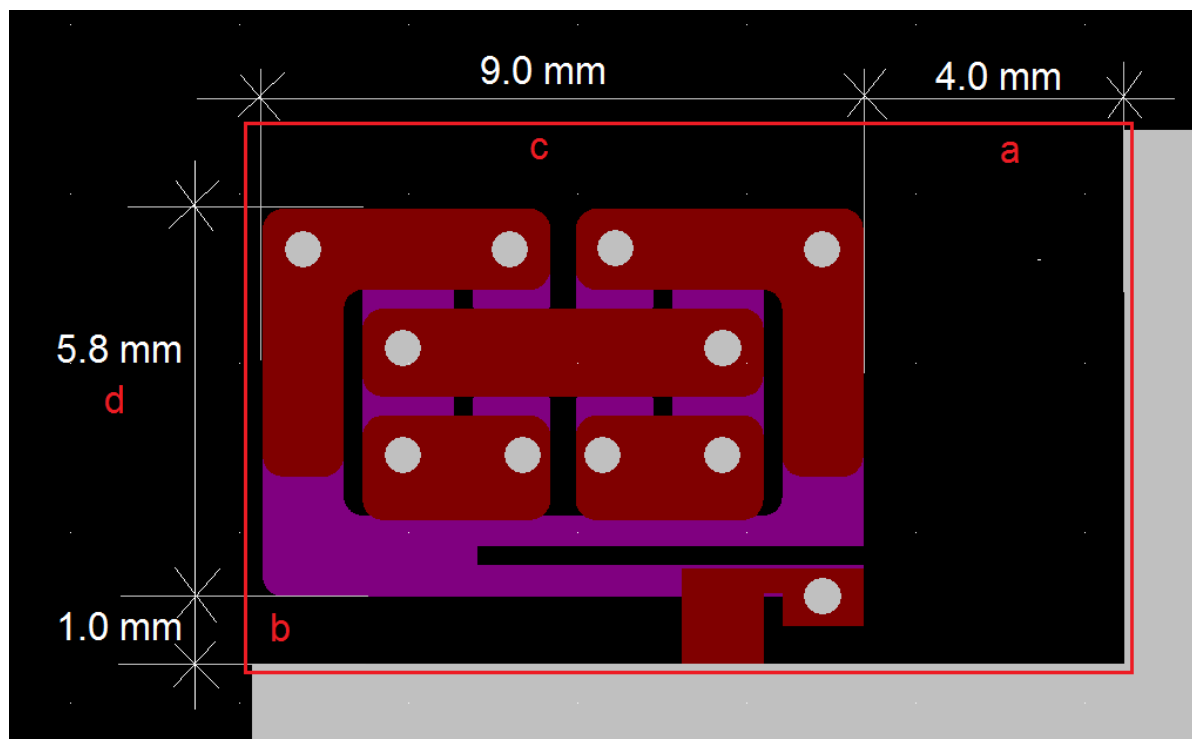
Figure 17: Radiation Pattern, PCB Vertical, Reduced Size IFA, Variant 2



## Designing Printed Antennas for Bluetooth® Low Energy

### 5.4 ZOR Antenna, 1.6 mm Substrate

This variant of the Zeroth Order Resonance (ZOR) antenna is designed for a substrate thickness of 1.6 mm and requires a minimum of two layers. The TOP and BOTTOM layers are used below.



**Figure 18: ZOR Antenna Preview**

The red outline indicates the antenna footprint, i.e. required allocation of PCB space. The antenna footprint is available on request in DXF format.

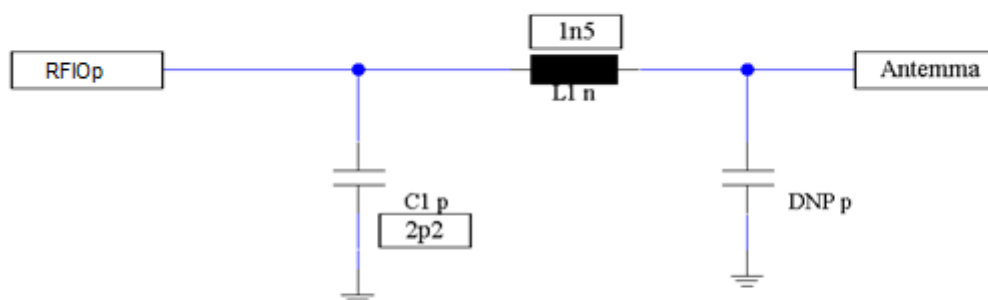
Legend (see [Figure 18](#)):

- a. Clearance between antenna and GND plane right.
  - i. 4.0 mm (157.5 mil).
  - ii. The GND on the right of the antenna is NOT required for correct antenna operation.
- b. Clearance between antenna and GND plane below.
  - i. 1.0 mm (39 mil).
- c. Antenna length
  - i. 9.0 mm (354 mil).
- d. Antenna width
  - i. 5.8 mm (228 mil).

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.4.1 Matching Network

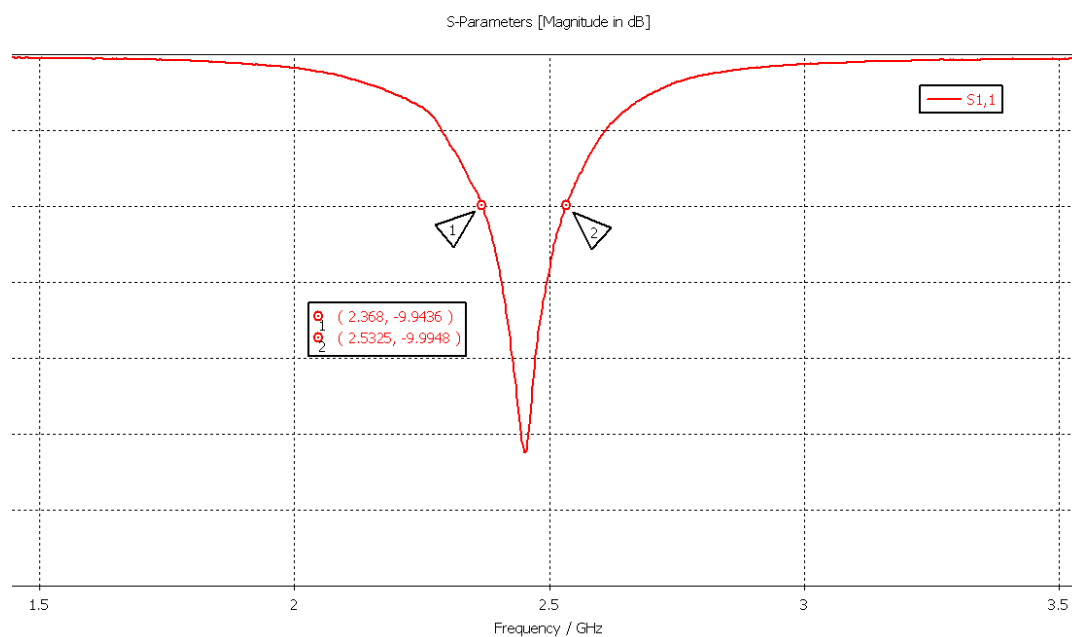
The matching is subject to change depending on substrate type or thickness and enclosure material (type and proximity to PCB).



**Figure 19: Matching Components, ZOR Antenna, 1.6 mm Substrate**

- L1: 1.5 nH, 0402, LQP series, Murata
- C1, 2.2 pF, 0402, GRM series, Murata
- In addition to the matching network, a 3.3 nH or 3.9 nH coil (depending on the DA1458x package) is required close to the RFIop pin.

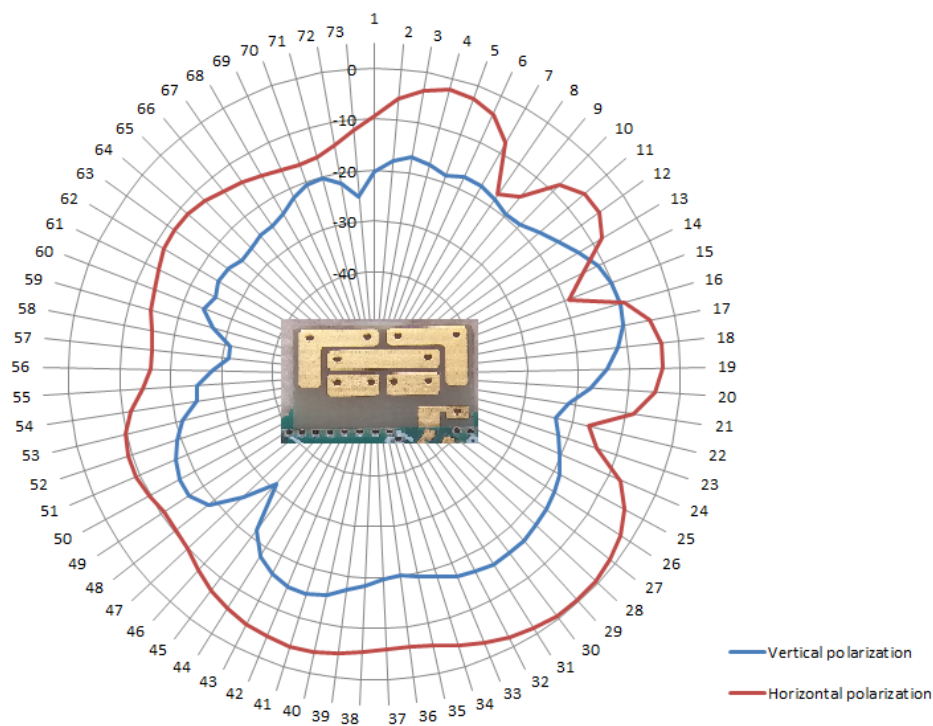
### 5.4.2 Measured Return Loss



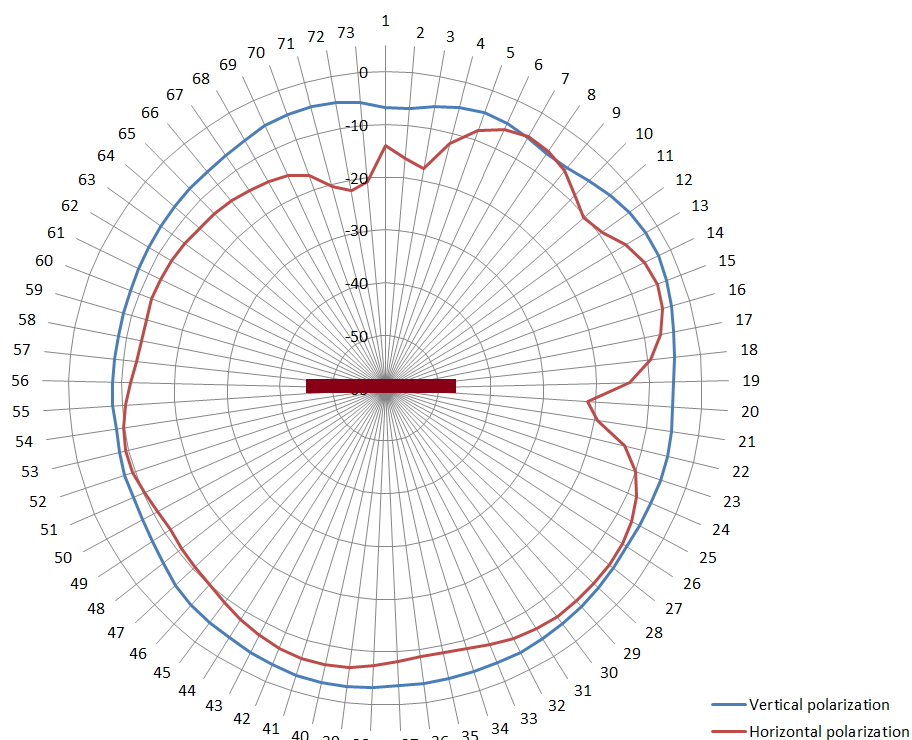
**Figure 20: Measured S11, ZOR Aantenna, 1.6 mm Substrate**

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.4.3 Measured Radiation Pattern



**Figure 21: Radiation Pattern, DUT Horizontal, ZOR Antenna, 1.6 mm Substrate**

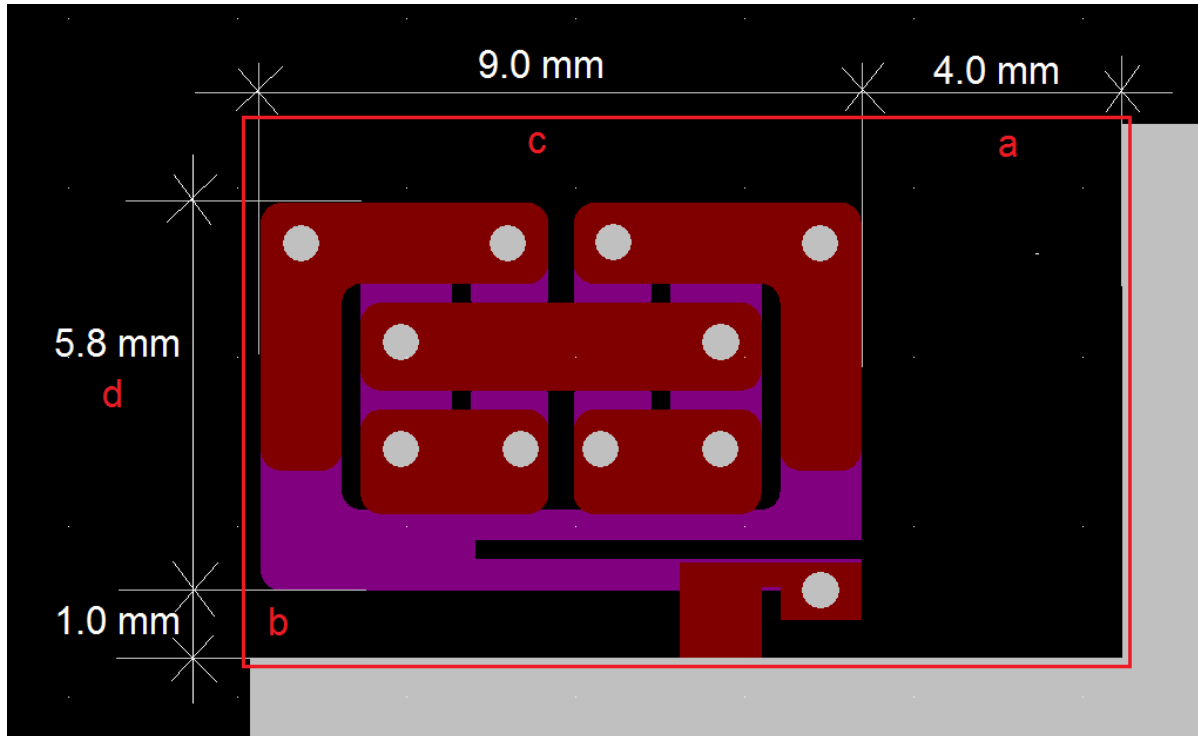


**Figure 22: Radiation Pattern, DUT Vertical, ZOR Antenna, 1.6 mm Substrate**

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.5 ZOR Antenna, 1.0 mm Substrate

This variant of the Zeroth Order Resonance (ZOR) antenna is designed for a substrate thickness of 1.0 mm and requires a minimum of two layers. The TOP and BOTTOM layers are used below.



**Figure 23: ZOR Antenna Preview**

The red outline indicates the antenna footprint, i.e. required allocation of PCB space. The antenna footprint is available on request in DXF format.

Legend (see [Figure 23](#)):

- a. Clearance between antenna and GND plane right.
  - i. 4.0 mm (157.5 mil).
  - ii. The GND on the right of the antenna is NOT required for correct antenna operation.
- b. Clearance between antenna and GND plane below.
  - i. 1.0 mm (39 mil).
- c. Antenna length
  - i. 9.0 mm (354 mil).
- d. Antenna width
  - i. 5.8 mm (228 mil).

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.5.1 Matching Network

The matching is subject to change depending on substrate type or thickness and enclosure material (type and proximity to PCB).

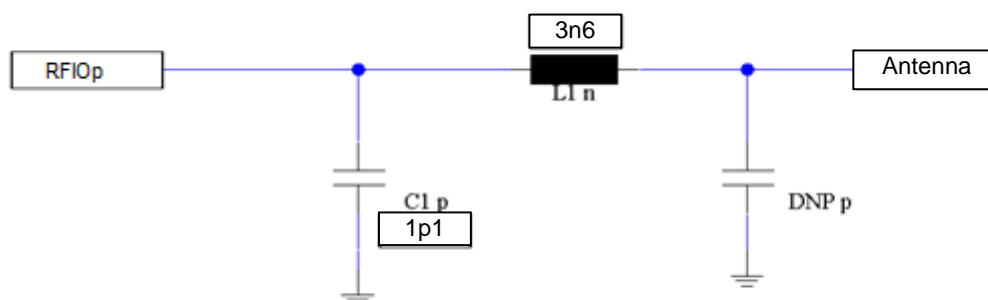


Figure 24: Matching Components, ZOR Antenna, 1.0 mm Substrate

- L1: 3.6 nH, 0402, LQP series, Murata
- C1, 1.1 pF, 0402, GRM series, Murata
- In addition to the matching network, a 3.3 nH or 3.9 nH coil (depending on the DA1458x package) is required close to the RFIO pin.

### 5.5.2 Measured Return Loss

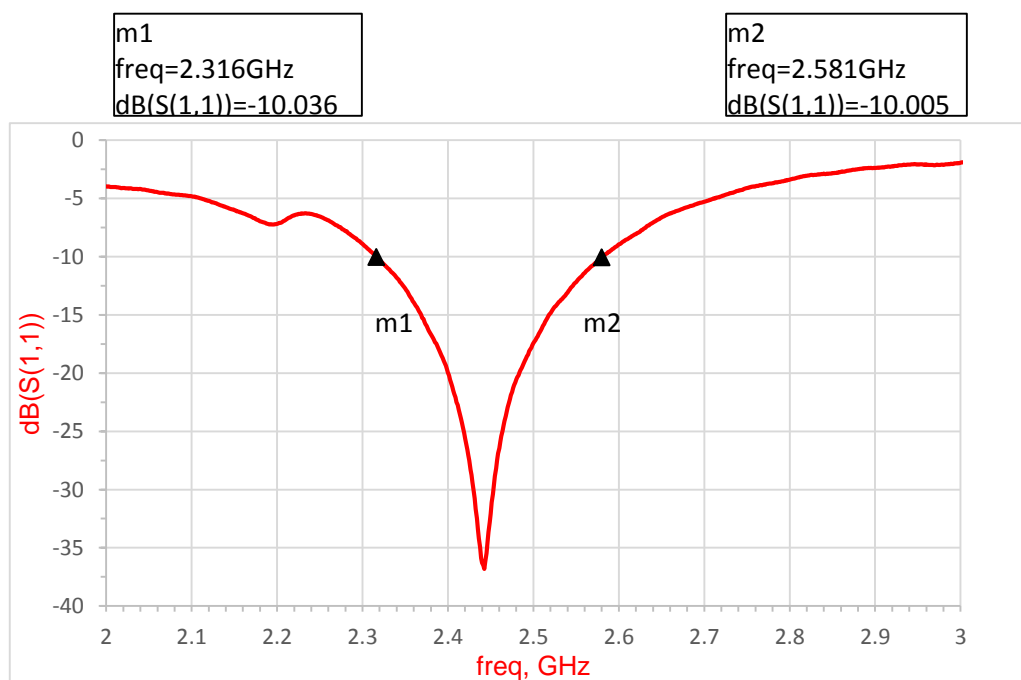


Figure 25: Measured S11, ZOR Antenna, 1.0 mm Substrate

## Designing Printed Antennas for Bluetooth® Low Energy

### 5.5.3 Measured Radiation Pattern

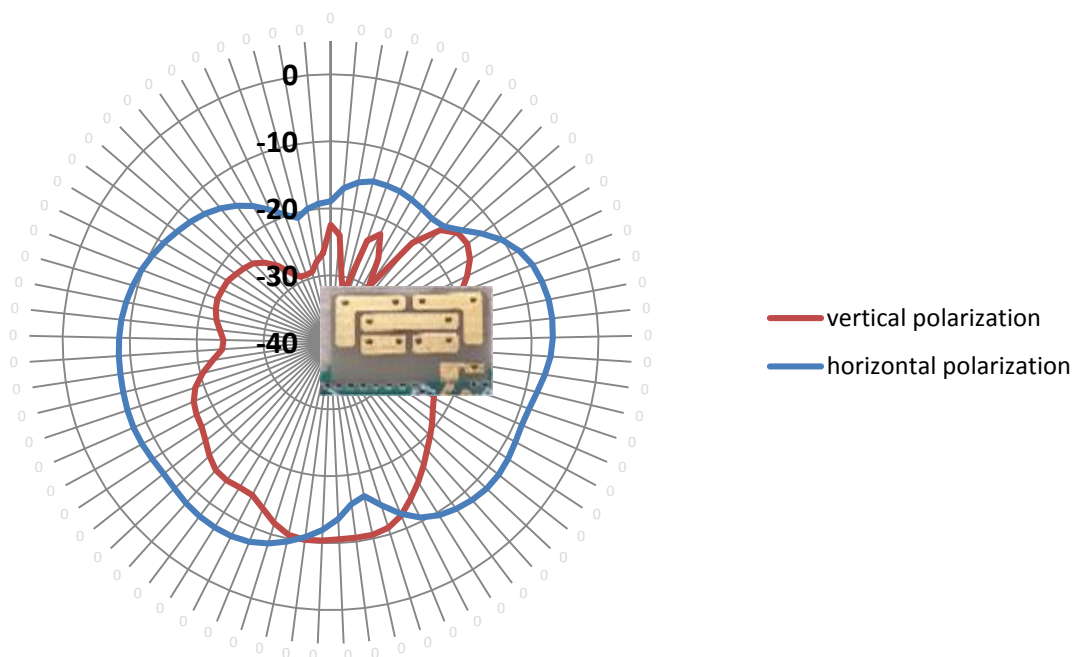


Figure 26: Radiation Pattern, DUT Horizontal, ZOR Antenna, 1.0 mm Substrate

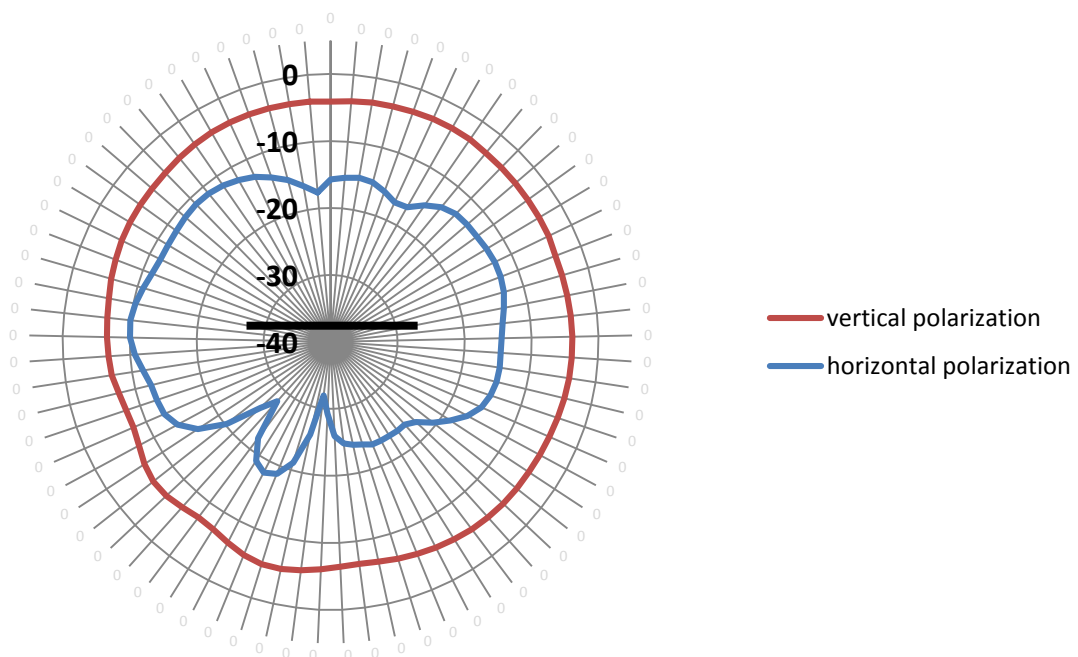


Figure 27: Radiation Pattern, DUT Vertical, ZOR Antenna, 1.0 mm Substrate

---

## Designing Printed Antennas for Bluetooth® Low Energy

---

### 6 Antennas on Single Layer Substrates

Single layer substrates require a different approach with regard to antennas.

Due to limited space and possibility to construct a continuous GND plane, printed dipoles are often used instead of IFAs. The dipole requires a balun when the radio input is single ended and therefore has additional costs in components.

Due to the single ended input of the DA1458x the IFA is still the best choice, but care must be taken that a proper GND plane is available. There are two main aspects to take care of:

- Continuous GND plane at the antenna (see section 6.2).
- Minimum GND plane size for correct antenna operation.

Because the antenna is implemented on one metal layer, the size of the antenna is slightly larger than the equivalent multi-layer design.

Single layer substrates are usually used to reduce costs and have a lower RF performance due to higher dielectric losses. This has a direct impact on the radiation efficiency. A typical IFA on an FR1 substrate can have a radiation efficiency between -2.5 dB and -5 dB.

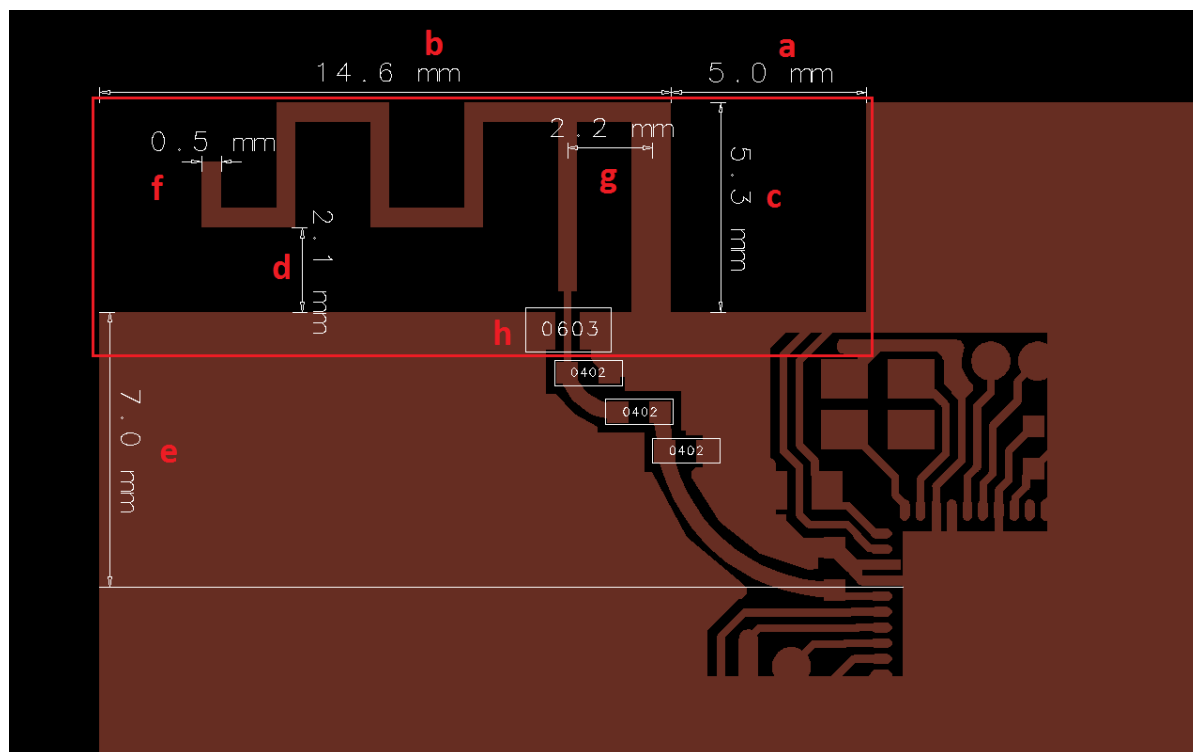
#### 6.1 Carbon Layers

Single layer PCBs are often used for low cost remote controls. It is common to use a carbon layer for the keyboard matrix or to have carbon bridges to route signals. The following rules must be applied to ensure maximum RF performance:

- Do not route RF signals in carbon.
- Do not cross RF transmission lines with carbon bridges.
  - An exception can be made when the carbon layer is on the other side of the PCB (i.e. copper on TOP and carbon on BOTTOM).
- Do not place carbon under the antenna.
  - The carbon is a conductive high loss material and will act as an absorber, thereby drastically reducing the radiation efficiency and the range of the antenna.

## Designing Printed Antennas for Bluetooth® Low Energy

### 6.2 Single Layer Printed IFA, 1 mm Substrate



**Figure 28: Single Layer IFA**

The dimensions above are given for a typical FR1 PCB substrate, 1 mm thick. The antenna length is adjusted for resonance, including a 1 mm plastic enclosure placed in contact with the PCB antenna.

The red outline indicates the antenna footprint, i.e. required allocation of PCB space. The antenna footprint is available on request in DXF format.

Legend (see [Figure 28](#)):

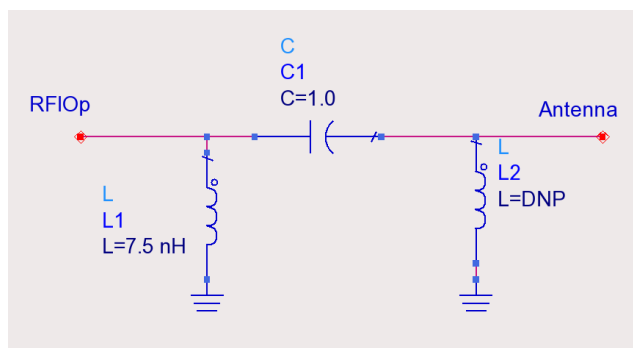
- a. Clearance between antenna arm and GND plane right.
- b. Antenna width.
- c. Antenna height.
- d. Clearance between the antenna arm and GND plane below.
- e. Minimum GND plane size required for correct operation of the antenna.
- f. Antenna trace width.
- g. Feed point position.
- h. 0  $\Omega$  0603 resistor used to connect the two sides of the GND planes.
  - i. When no DC connection is required, a 10 pF capacitor can also be used.
  - ii. This component is essential for the correct operation of the antenna.



## Designing Printed Antennas for Bluetooth® Low Energy

### 6.2.1 Matching Network

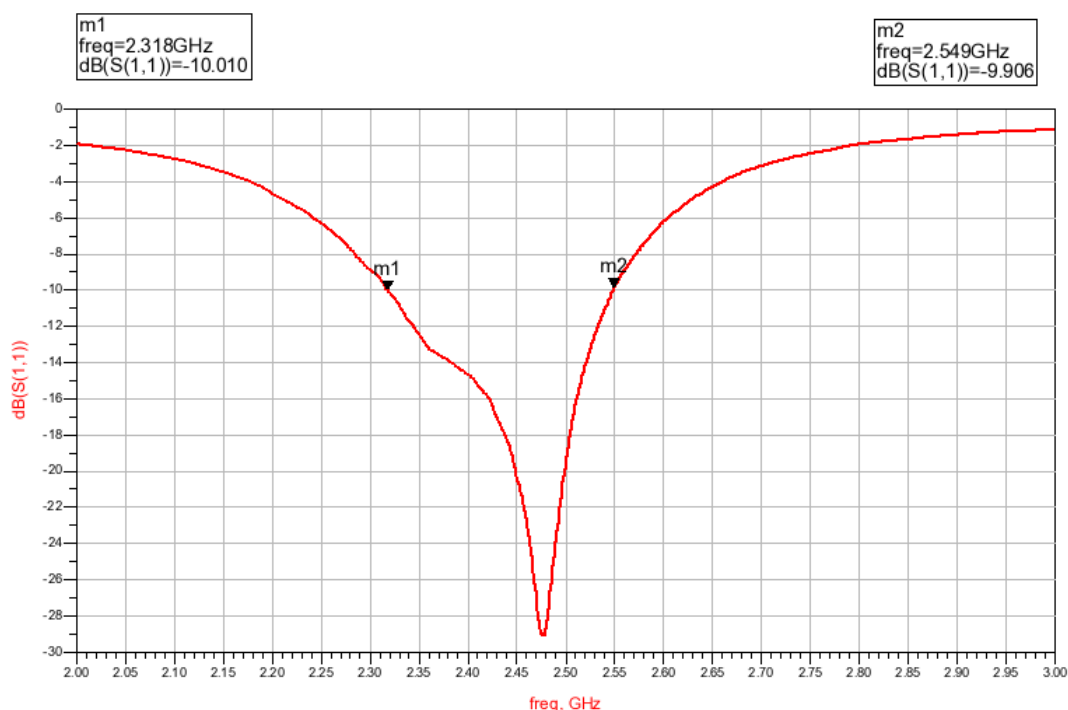
The matching is subject to change depending on substrate type or thickness and enclosure material (type and proximity to PCB).



**Figure 29: Matching Components, Single Layer IFA**

- L1: 7.5 nH, 0402, LQP series, Murata
- C1: 1.0 pF, 0402, GRM15 series, Murata
- L2: DNP
- In addition to the matching network, a 3.3 nH or 3.9 nH coil (depending on the DA1458x package) is required close to the RFIO pin.

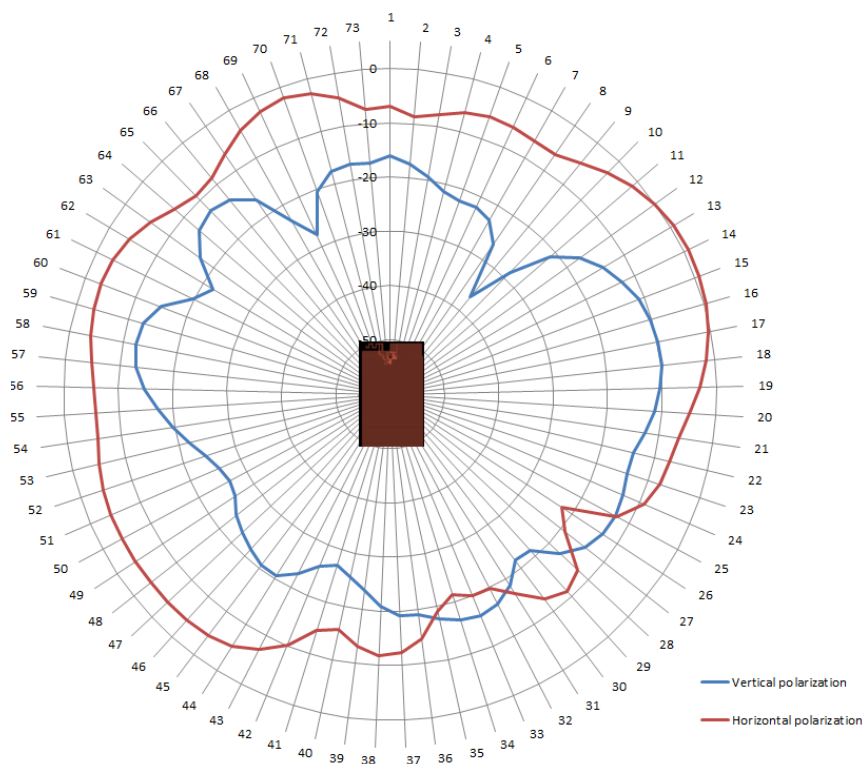
### 6.2.2 Measured Return Loss



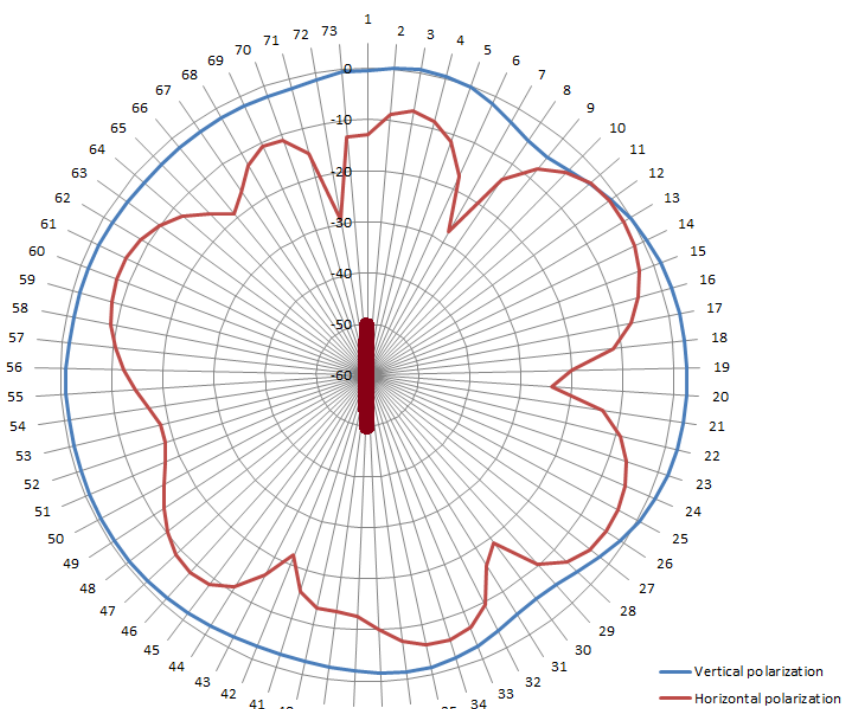
**Figure 30: Measured S11, Single Layer IFA**

## Designing Printed Antennas for Bluetooth® Low Energy

### 6.2.3 Measured Radiation Pattern



**Figure 31: Radiation Pattern, PCB Horizontal, Single Layer IFA**



**Figure 32: Radiation Pattern, PCB Vertical, Single Layer IFA**

---

## Designing Printed Antennas for Bluetooth® Low Energy

---

### 7 Conclusions

Antennas play an important role in the overall performance of any radio based equipment. The guidelines in this document will enable customers to integrate printed low cost antennas in their Bluetooth low energy products with minimum effort and RF knowledge.

The antennas were designed, tested and measured at Dialog Semiconductor and are currently used in many of our customers' products.

---

**Designing Printed Antennas for Bluetooth® Low Energy**

---

**8 Revision History**

Revision	Date	Description
1.0	29-Aug-2014	Initial version.
2.0	26-Jul-2016	Added ZOR antenna, 1.6 mm substrate (section <a href="#">5.4</a> ).
2.1	04-Jan-2017	Added ZOR antenna, 1.0 mm substrate (section <a href="#">5.5</a> ).
2.2	22-Feb-2017	Generalizing the document for DA1458x family.

## Designing Printed Antennas for Bluetooth® Low Energy

### Status Definitions

Status	Definition
DRAFT	The content of this document is under review and subject to formal approval, which may result in modifications or additions.
APPROVED or unmarked	The content of this document has been approved for publication.

### Disclaimer

Information in this document is believed to be accurate and reliable. However, Dialog Semiconductor does not give any representations or warranties, expressed or implied, as to the accuracy or completeness of such information. Dialog Semiconductor furthermore takes no responsibility whatsoever for the content in this document if provided by any information source outside of Dialog Semiconductor.

Dialog Semiconductor reserves the right to change without notice the information published in this document, including without limitation the specification and the design of the related semiconductor products, software and applications.

Applications, software, and semiconductor products described in this document are for illustrative purposes only. Dialog Semiconductor makes no representation or warranty that such applications, software and semiconductor products will be suitable for the specified use without further testing or modification. Unless otherwise agreed in writing, such testing or modification is the sole responsibility of the customer and Dialog Semiconductor excludes all liability in this respect.

Customer notes that nothing in this document may be construed as a license for customer to use the Dialog Semiconductor products, software and applications referred to in this document. Such license must be separately sought by customer with Dialog Semiconductor.

All use of Dialog Semiconductor products, software and applications referred to in this document are subject to Dialog Semiconductor's [Standard Terms and Conditions of Sale](http://www.dialog-semiconductor.com), available on the company website ([www.dialog-semiconductor.com](http://www.dialog-semiconductor.com)) unless otherwise stated.

Dialog and the Dialog logo are trademarks of Dialog Semiconductor plc or its subsidiaries. All other product or service names are the property of their respective owners.

© 2017 Dialog Semiconductor. All rights reserved.

## Contacting Dialog Semiconductor

### United Kingdom (Headquarters)

*Dialog Semiconductor (UK) LTD*  
Phone: +44 1793 757700

### Germany

*Dialog Semiconductor GmbH*  
Phone: +49 7021 805-0

### The Netherlands

*Dialog Semiconductor B.V.*  
Phone: +31 73 640 8822

### Email:

[enquiry@diasemi.com](mailto:enquiry@diasemi.com)

### North America

*Dialog Semiconductor Inc.*  
Phone: +1 408 845 8500

### Japan

*Dialog Semiconductor K. K.*  
Phone: +81 3 5425 4567

### Taiwan

*Dialog Semiconductor Taiwan*  
Phone: +886 281 786 222

### Web site:

[www.dialog-semiconductor.com](http://www.dialog-semiconductor.com)

### Singapore

*Dialog Semiconductor Singapore*  
Phone: +65 64 8499 29

### Hong Kong

*Dialog Semiconductor Hong Kong*  
Phone: +852 3769 5200

### Korea

*Dialog Semiconductor Korea*  
Phone: +82 2 3469 8200

### China (Shenzhen)

*Dialog Semiconductor China*  
Phone: +86 755 2981 3669

### China (Shanghai)

*Dialog Semiconductor China*  
Phone: +86 21 5424 9058