5. Antennas on Single Layer Substrates

Single layer substrates require a different approach regarding 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 Bluetooth LE device, 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 5.2).
- 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 cost but have a lower RF performance due to higher transmission losses. This has a direct impact on the radiation efficiency. A typical IFA on a single layer substrate has a radiation efficiency between -2.5 dB and -5 dB.

5.1 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 (that is copper on TOP and carbon on BOTTOM).
- Do not place carbon under the antenna.
  - The carbon is a conductive high loss material and acts as an absorber, thereby drastically reducing the radiation efficiency and the range of the antenna.

5.2 Single Layer Printed IFA, 1 mm Substrate

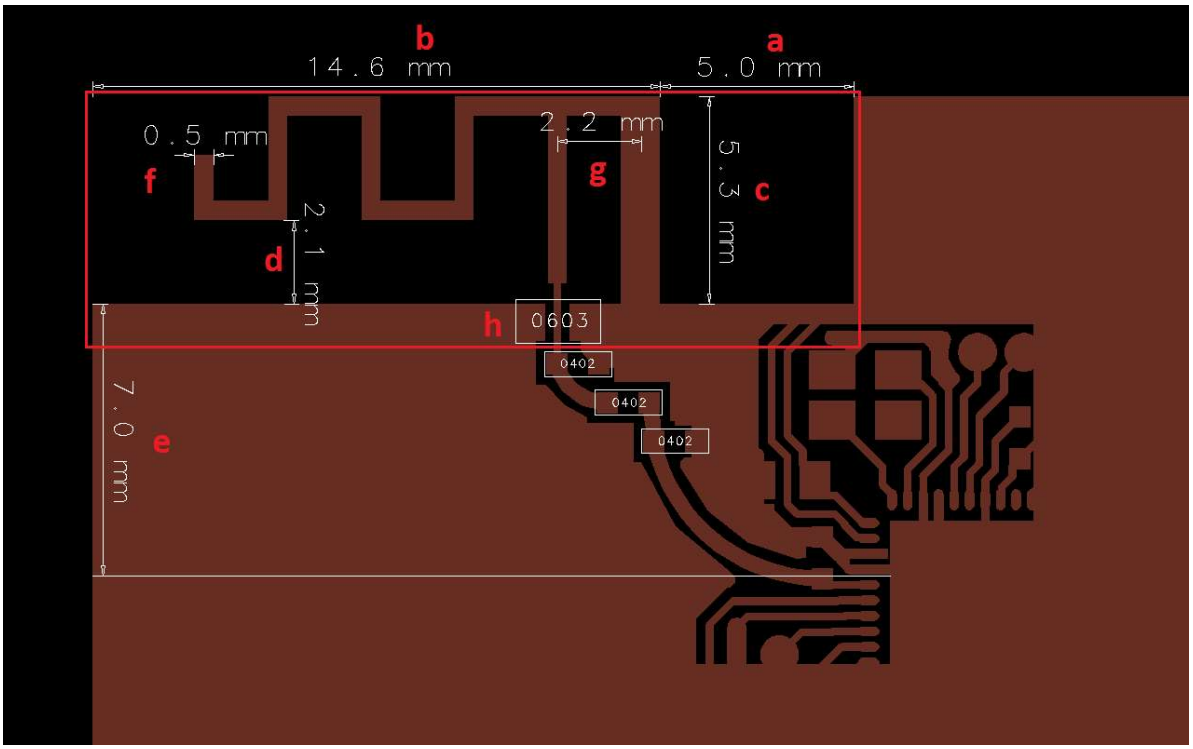

Figure 28. Single layer IFA

The dimensions above are given for a typical single layer 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, that is the 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 Ω 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.

i. Antenna model: CFR0014

j.Bluetooth operating frequency: 2402-2480MHz

k. Antenna Gain: 0dBi

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).

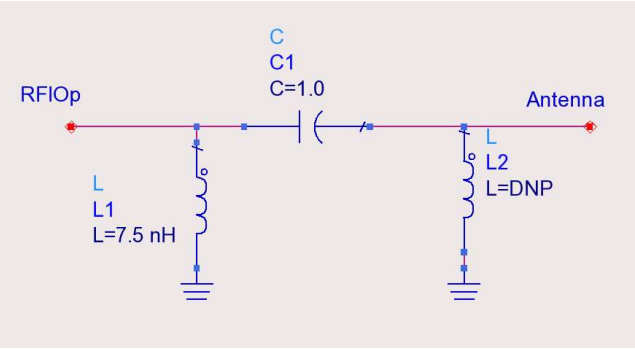

Figure 29. Matching components, single layer IFA

- 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 Bluetooth LE devicepackage) is required close to the RFIOp pin.

5.2.2 Measured Return Loss

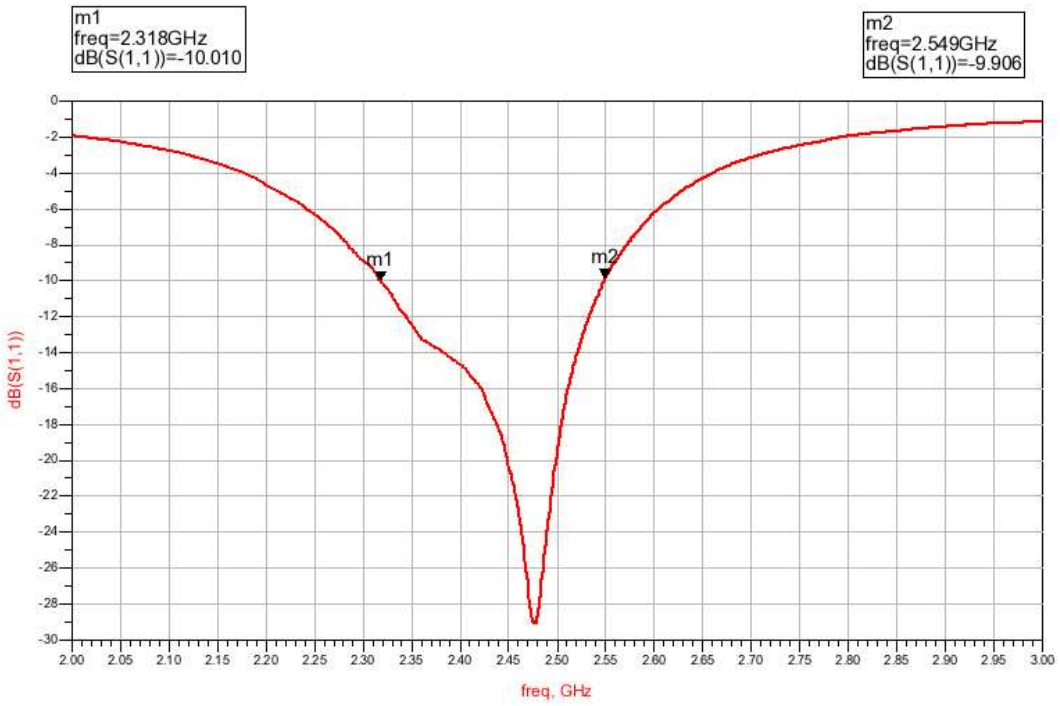

Figure 30. Measured S11, single layer IFA

5.2.3 Measured Radiation Pattern

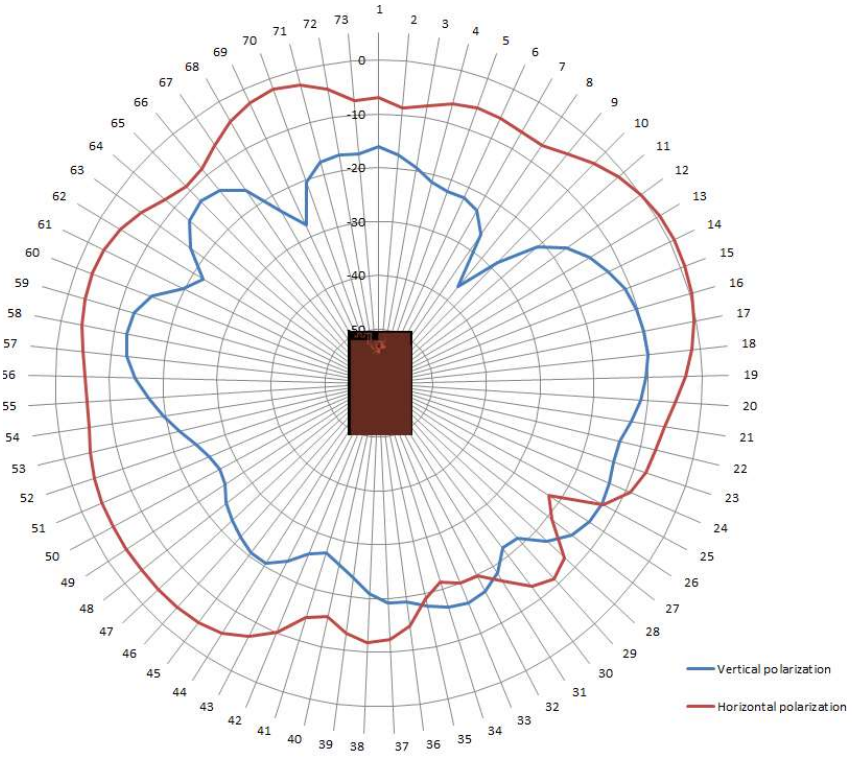

Figure 31. Radiation pattern, PCB horizontal, single layer IFA

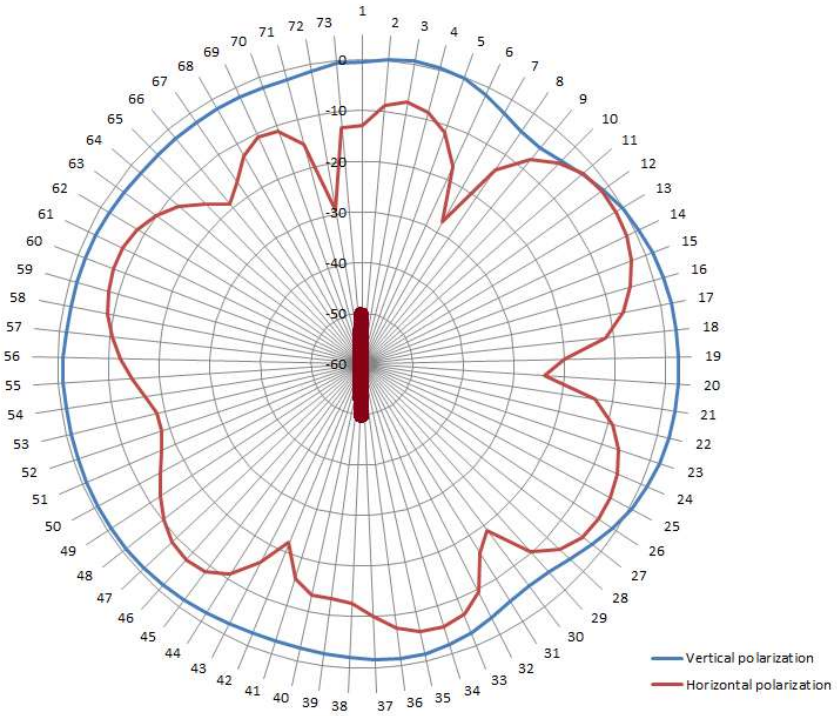

Figure 32. Radiation pattern, PCB vertical, single layer IFA