

Designing Printed Antennas for Bluetooth® Low Energy

5.1 Full Size Printed IFA, 1 mm Substrate

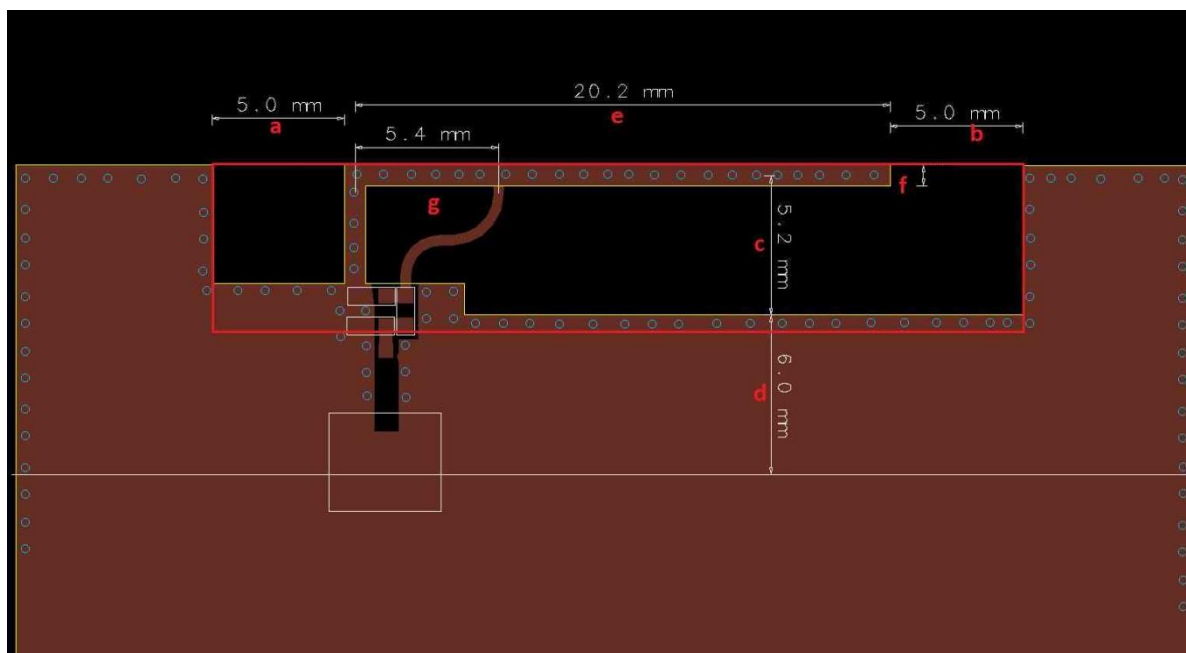


Figure 3: Full Size IFA with Dimensions

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).

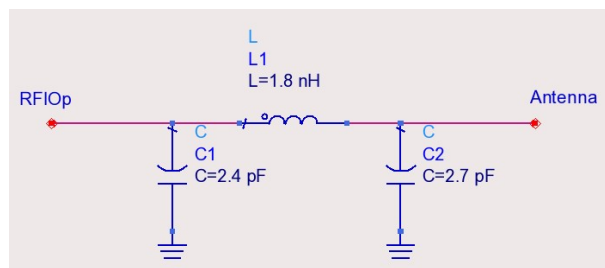


Figure 4: Matching Components, Full Size IFA

- 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

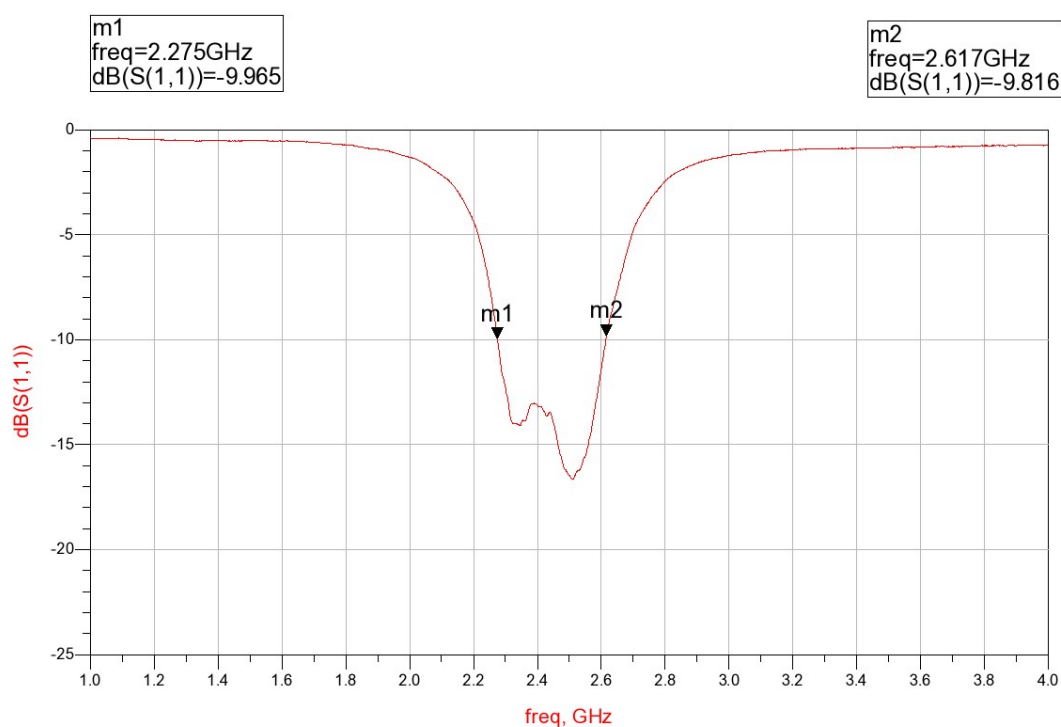


Figure 5: Measured S11, Full Size IFA

Designing Printed Antennas for Bluetooth® Low Energy

5.1.3 Measured Radiation Pattern

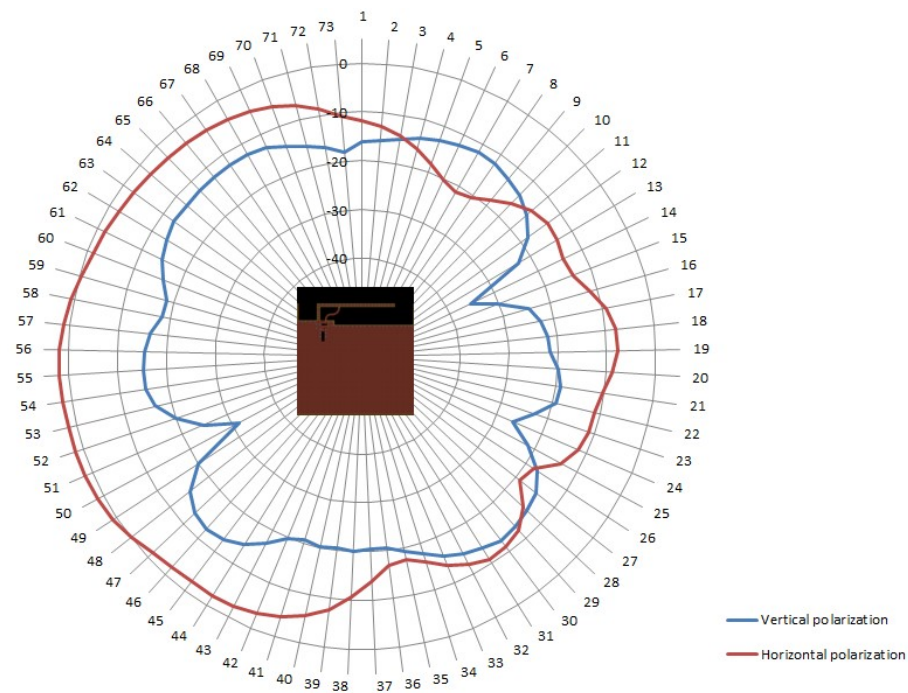


Figure 6: Radiation Pattern, PCB Horizontal, Full Size IFA

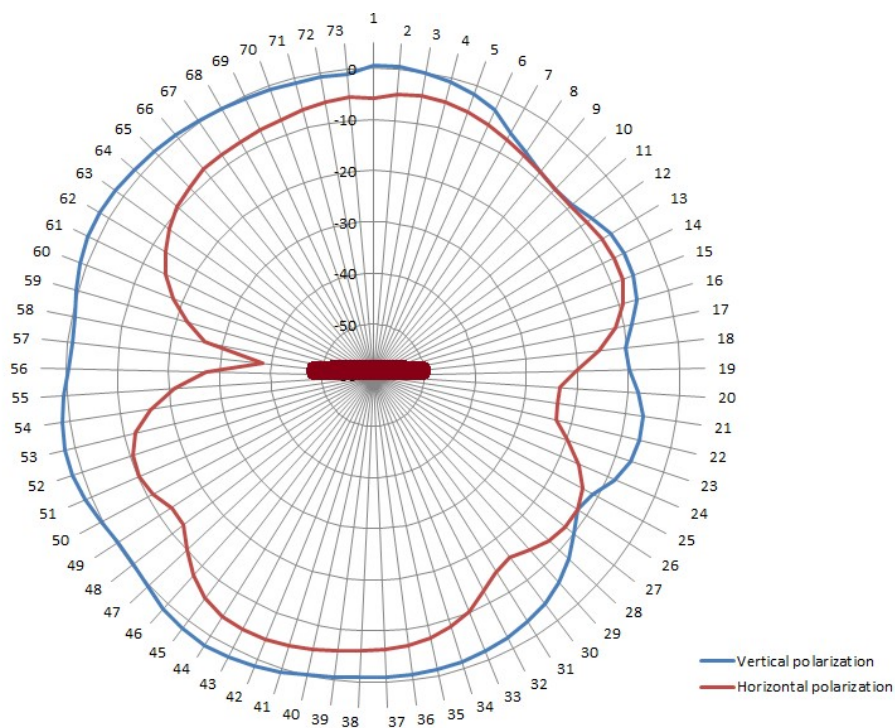


Figure 7: Radiation Pattern, PCB Vertical, Full Size IFA