FOR Smart Card : BobeePass

# Specifications

## Information on the equipment

| | |
|---|---|
| Device name | PCB Antenna |
| Brand | SMARTdisplayer |
| Model No. | BobeePass |
| Manufacturer | Dialog Semiconductor GmbH |
| Operating Frequency | 2.4GHZ |
| Gain | 2.09dBi |
| Receiving bandwidth | 100MHZ-150MHZ |

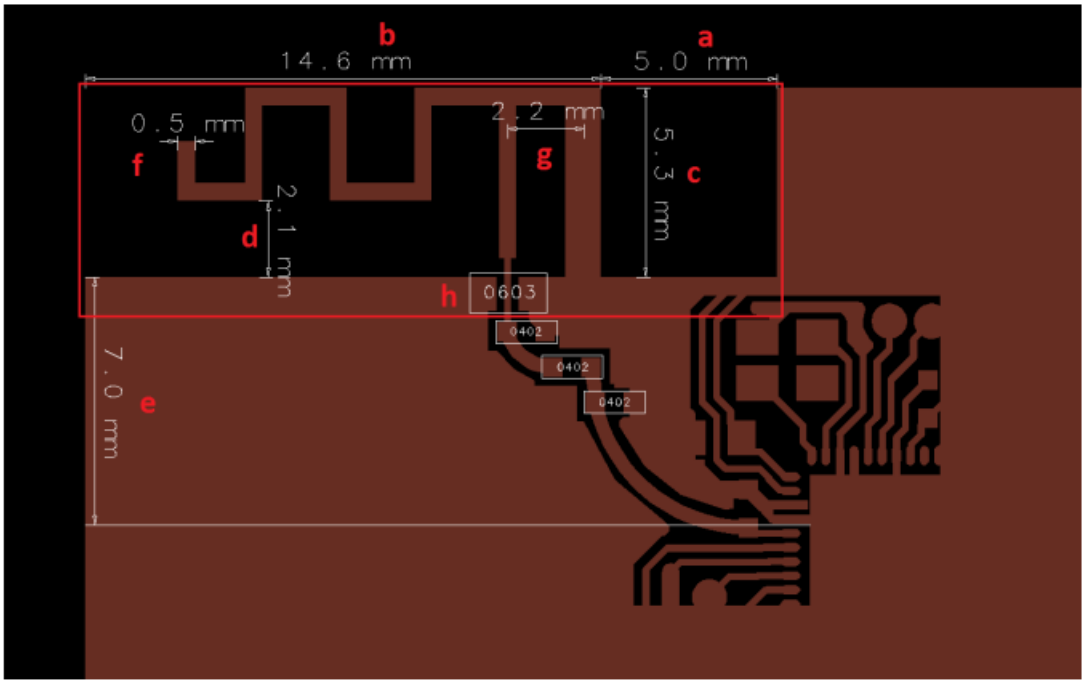


**Figure 18: Single layer IFA**

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

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

Legend (see Figure 18):

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.

# 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 DA14580 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.

### 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).*

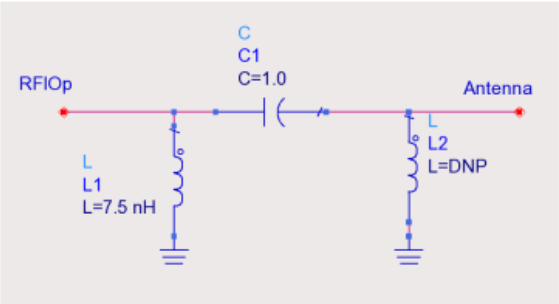


**Figure 19: Matching components, single layer IFA**

- L1: 7.5 nH, 0402, LQP series, Murata
- C1: 1.0 pF, 0402, GRM15 series, Murata
- L2: DNP
- *In addition to the matching a network, a 3.3 nH or 3.9 nH coil (depending on the DA14580 package) is additionally required close to the RFIOp pin.*

### 6.2.3 Measured radiation pattern

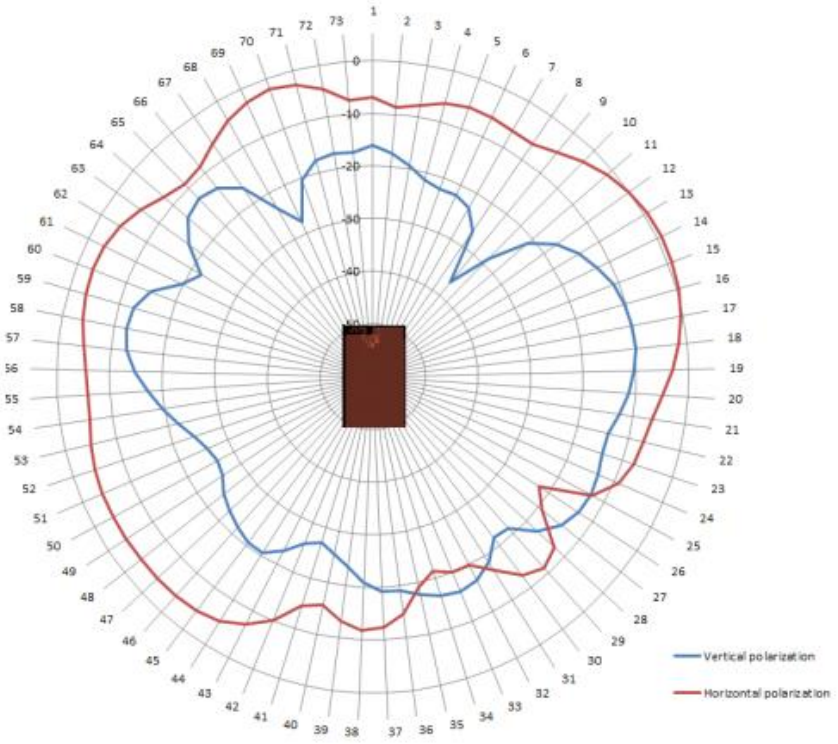


**Figure 21: Radiation pattern, PCB horizontal, single layer IFA**

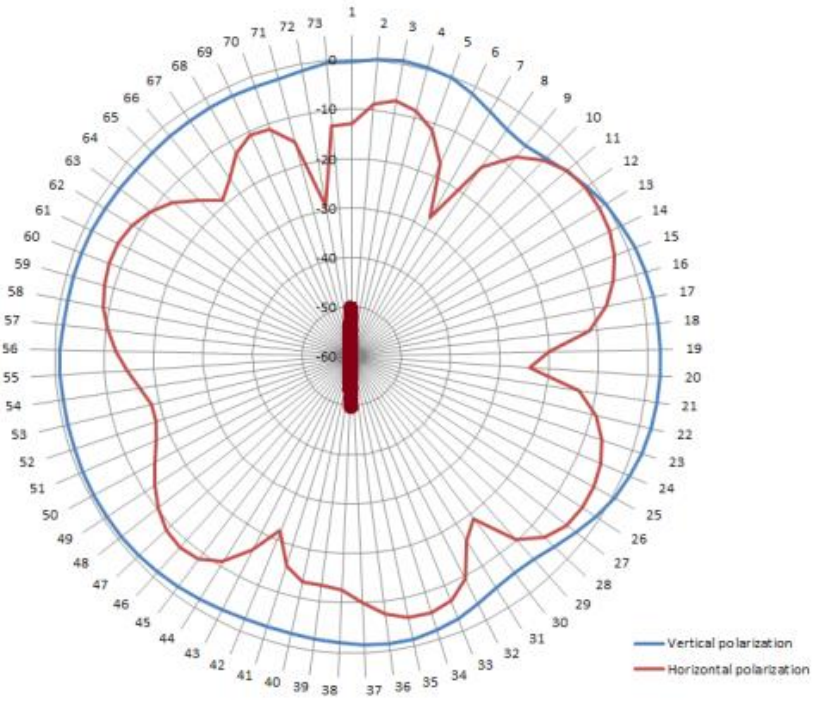


**Figure 22: Radiation pattern, PCB vertical, single layer IFA**