Magic ID Design Guideline (AN0198)

V1.0 – May 13, 2019

No.19, Industry E. Rd. IV, Hsinchu Science Park, Hsinchu City 30077, Taiwan, R.O.C

TEL:+886-3-6662118 FAX:+886-3-6662117 Http://www.generalplus.com

1 Introduction

Magic reader, a low cost ID reorganization technology, is developed based on electric field communication. As shown in Figure 1, reader is a transmitter pad surrounded by multiple receiver pads. The transmitter will emit small amount of electric field signal while in the sensing phase. If there is no “path” or “bridge” inside the tag or card, pure electric field will decay quickly and receiver pads get nothing. When we put a card on the top of the reader, the connection between the transmitter pad and receiver pads provide the electric field paths. These paths can make receiver pads get enough electric field and confirm these connections, as shown in Figure 2. With different combinations, N receiver pads is able to form up to $(2^N - 1)$ unique IDs.

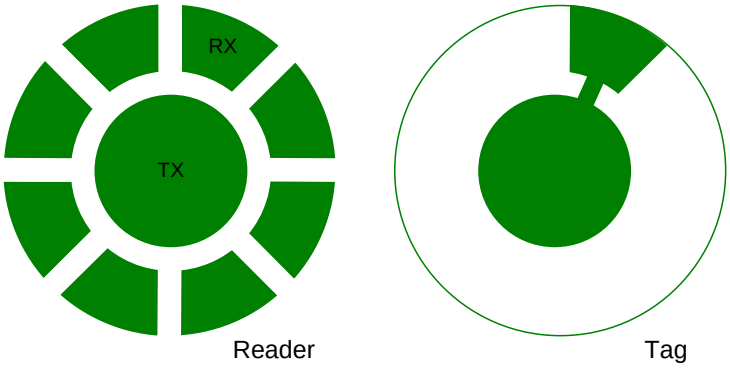

Figure 1

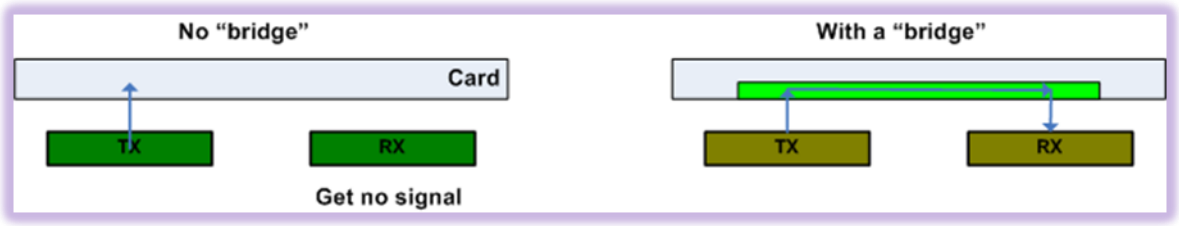

Figure 2

2 Magic Reader Structure

2.1 Reader

2.1.1 Various shape and arrangement

The pad’s shape, size, and arrangement may be various according to user’s demands. Figure 3 depicts some typical reader’s patterns.

It is also possible to form these tags into groups, as shown in Figure 4. Those three tags have the same connections to RX4 and RX5, but different TX pads make those tags different meanings. There are many possibilities which allow users to design and meet various requirements for variety of application needs.

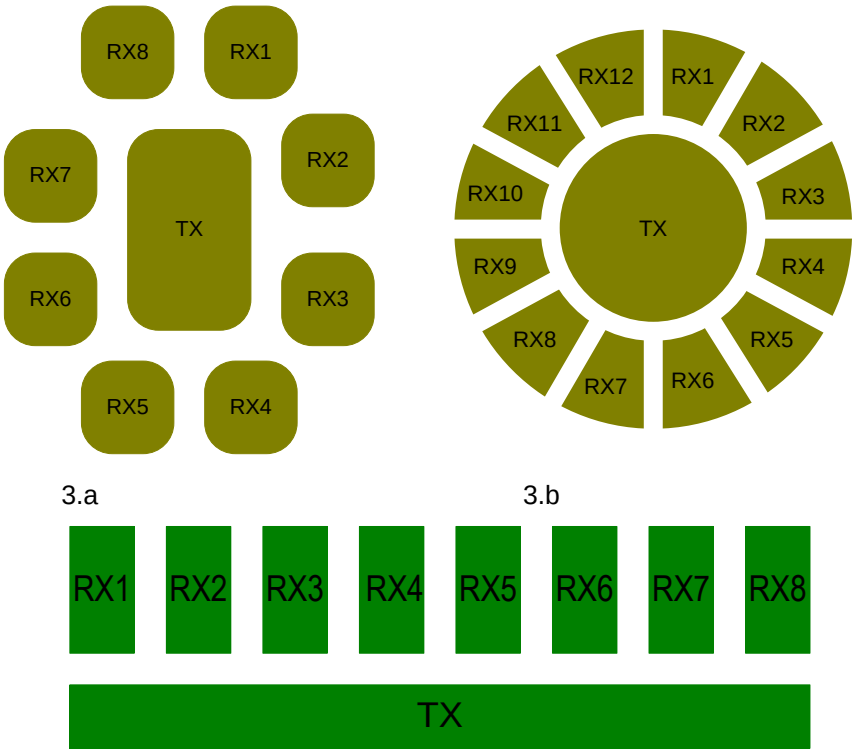

Figure 3

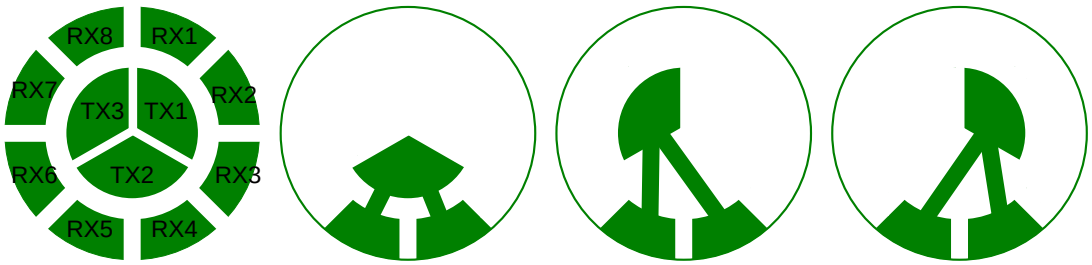

Figure 4

2.1.2 Material

It is recommended using the standard PCB for a reader. Membrane is also a type of solution, but due to its high impedance, layout should be more carefully when using membrane. Please refer to layout section for more information.

2.1.3 Size

Size is the important part of magic ID application. Ideally, larger pad provides better communication capability. However, small tags may be required in some cases. According to our experiences, the minimum diameter for a round shape reader is approximately 25mm.

Figure 5 is a reference design for a round shape pad. Moreover, size also matters when the thickness of a tag or mechanism's overlay is thick. All of these factors must be taken into account when designing an application.

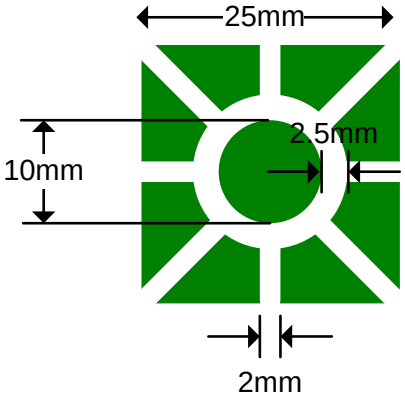

Figure 5

2.2 Tag

2.2.1 Shape

The shape of a tag usually follows its reader. Cover area between reader and tag is crucial for coupling electric field. User can make it a little bit smaller to allow some alignment tolerances but not too small. Please refer to Figure 6.

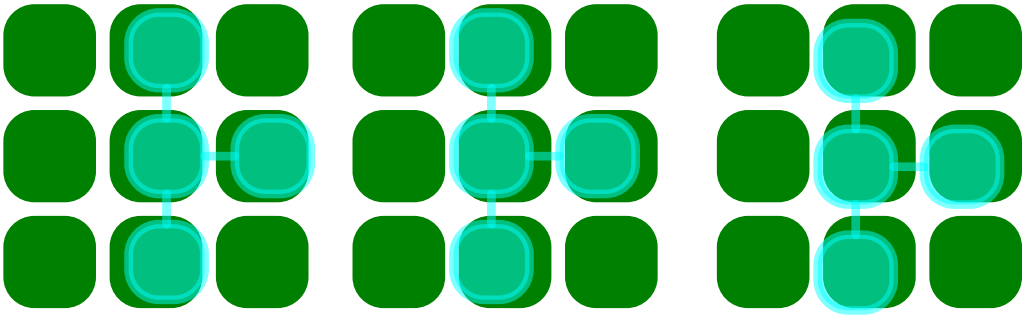

Figure 6

2.2.2 Layout

Figure 7 and Figure 8 show some combinations inside the tag. Leave the unused pads and their wires empty to prevent coupling noise. For example, an 8-pad tag can have ID from "00000001" ~ "11111111", total of 255 IDs available. "00000000" is meaningless in this technology.

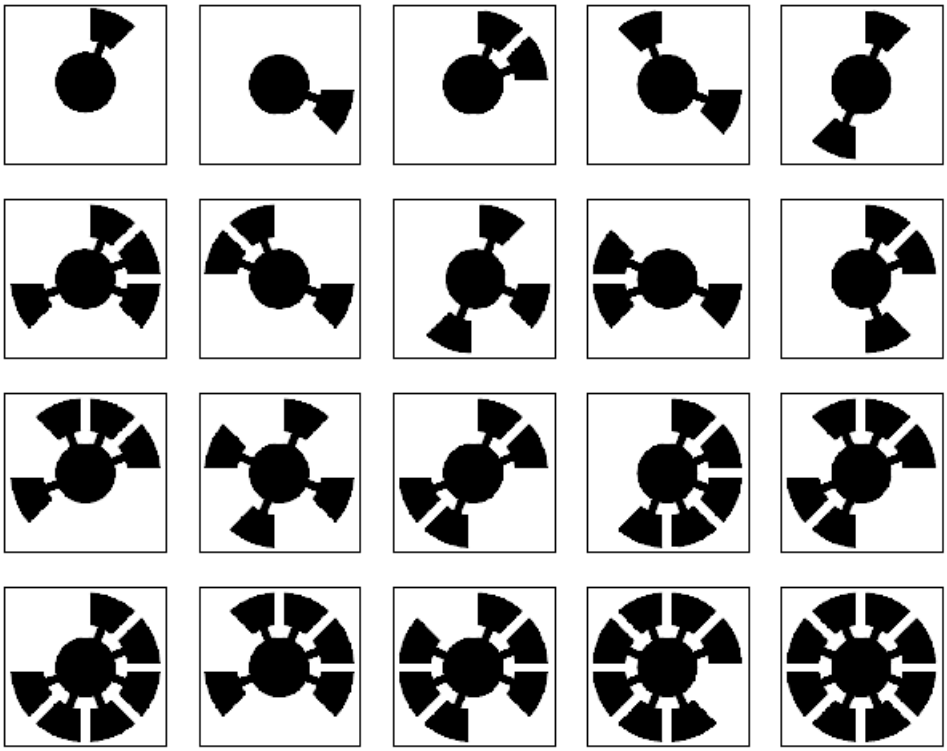

Figure 7

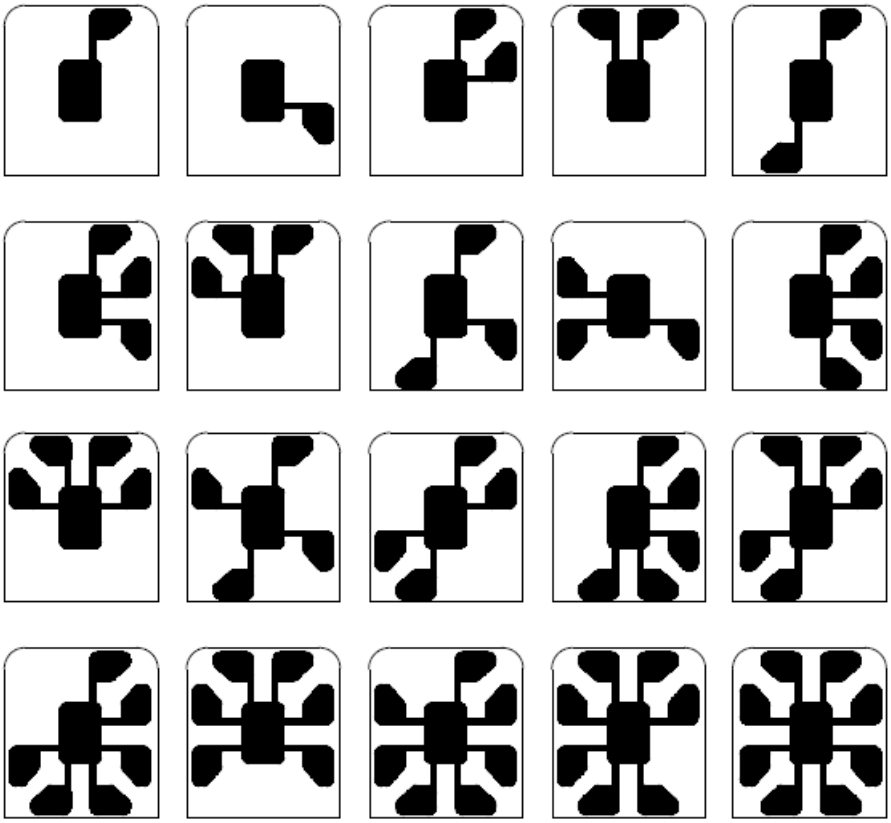

Figure 8

2.2.3 Material

Inside the tag could be many kinds of conductive materials such as copper, carbon, conductive ink, etc. Considering the paths inside a tag is used to pass the electric field signals. A low impedance material could pass more electric field signals than those materials with high impedance; thus, it makes the detection more stable and reliable.

3 Overlay

A common structure for magic reader is a sandwich type, as shown in Figure 9. Reader PCB is at the bottom, mechanism overlay or cover in the middle, and the tag on the top. From the electric field formula, $\mathbf{E} = \frac{1}{4\pi\varepsilon}\frac{Q}{d^2}$, we know it decays proportionally to d^2 where d is the distance between PCB and tag. Therefore, the sensitivity is low when overlay thickness is thicker, shown in Figure 10.

Figure 9

Based on our analysis, try not to let the thickness greater than 2mm. It is recommended having this overlay less than 1mm, thinner the better. If a thicker overlay is not avoidable, use a better conductive material inside the tag.

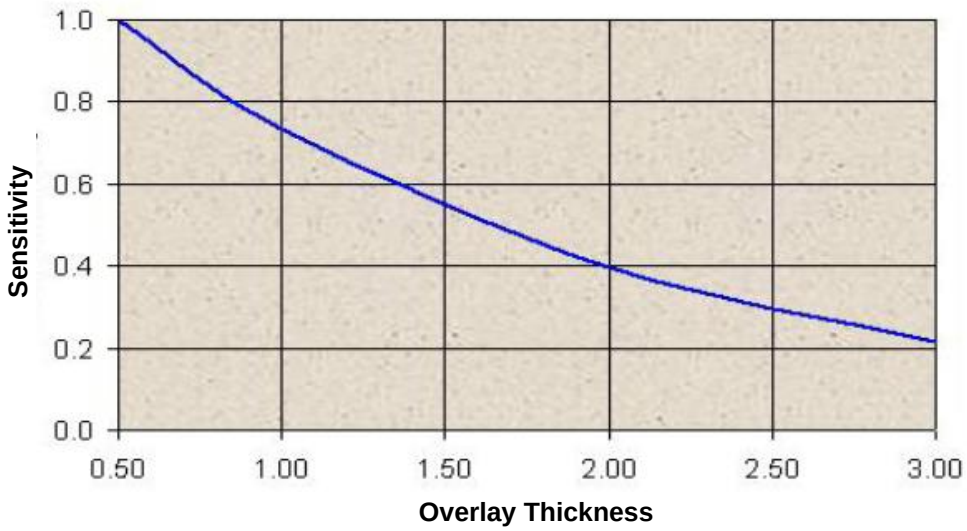

Figure 10

4 Matrix type

Magic reader also supports matrix type detection. Figure 11 is a 4 x 8 matrix application. Suppose there are N transmitters and M receivers, Magic reader can provide N detection positions and each can detect $2^M - 1$ IDs. More transmitters and receivers result in longer detection time, meaning lower response time.

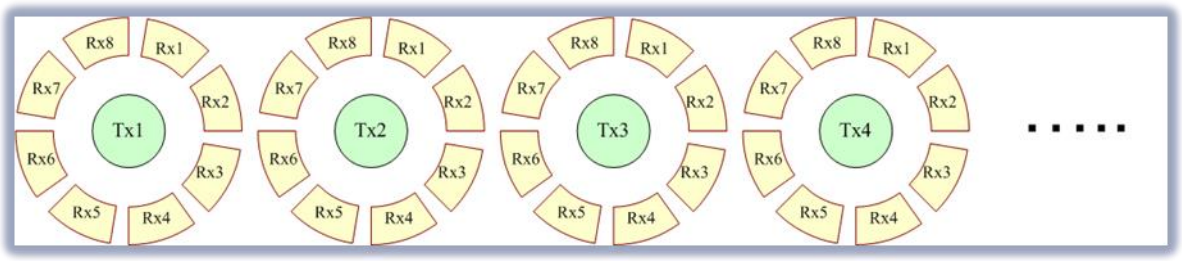

Figure 11

5 Different Sensing Schematic

5.1 Direct Drive

A thinner overlay or larger pad size may be the most economical solution. Each receiver pad is pulled-up by a 4M ~ 6M-Ohm resistor. Transmitter pad is directly connected to an IO.

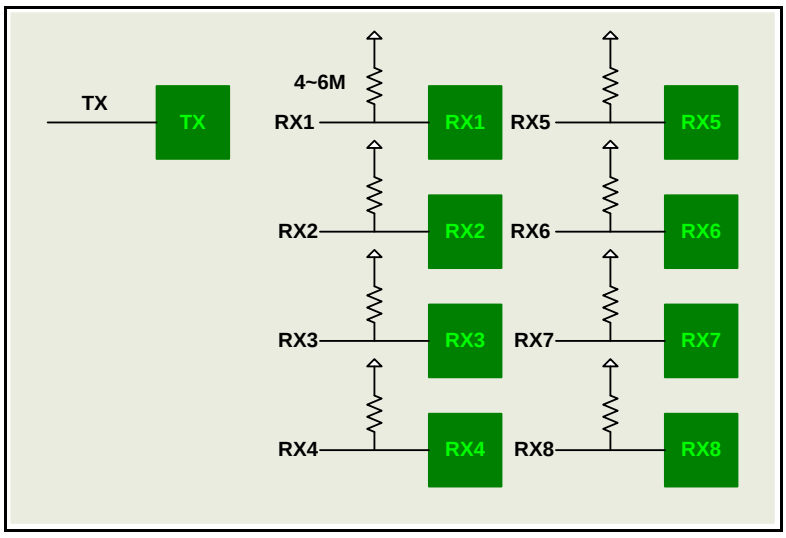

Figure 12

5.2 Double voltage boost

Suppose the overlay is thicker and there are multiple transmitter pads required in an application. Thus, double voltage boost circuit is a suitable solution for this type of application. Note that PWM IO and Vdd in Figure 13 should be lower than 3.3V. Assume all the VDDs in the figure are 3.3V, Vout is around 5.2V ~ 5.8V according to the forward voltage Vf of diode. Use Vout as the power source of VddIO of TX and still use Vdd for VddIO of RX and its pull-up voltage.

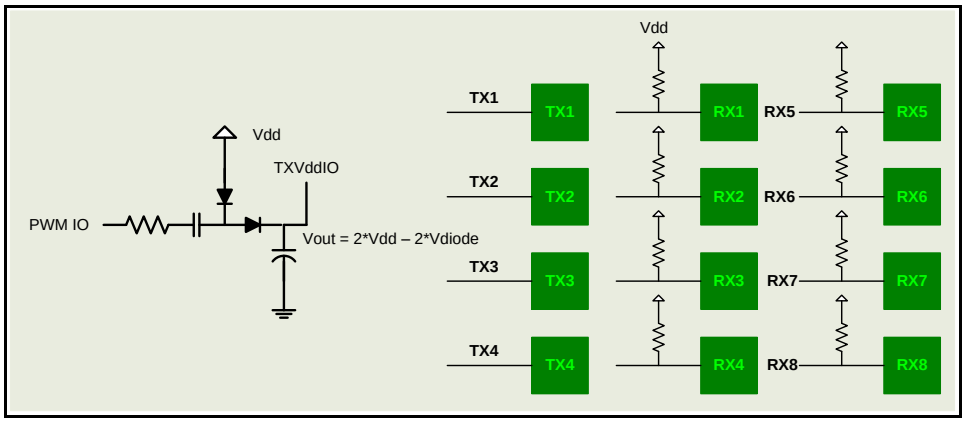

Figure 13

5.3 Inductor boost

If the overlay is thick and only few transmitter pads will be equipped, Figure 12 is an option to apply. By properly choosing the PWM frequency, PWM duty, and inductor, user can control the pump voltage from 10V to 30V. Through a simple transistor switch, transmitter pad is able to provide better electric field. Under this condition, it is able to have 2mm ~ 3mm detection distance.

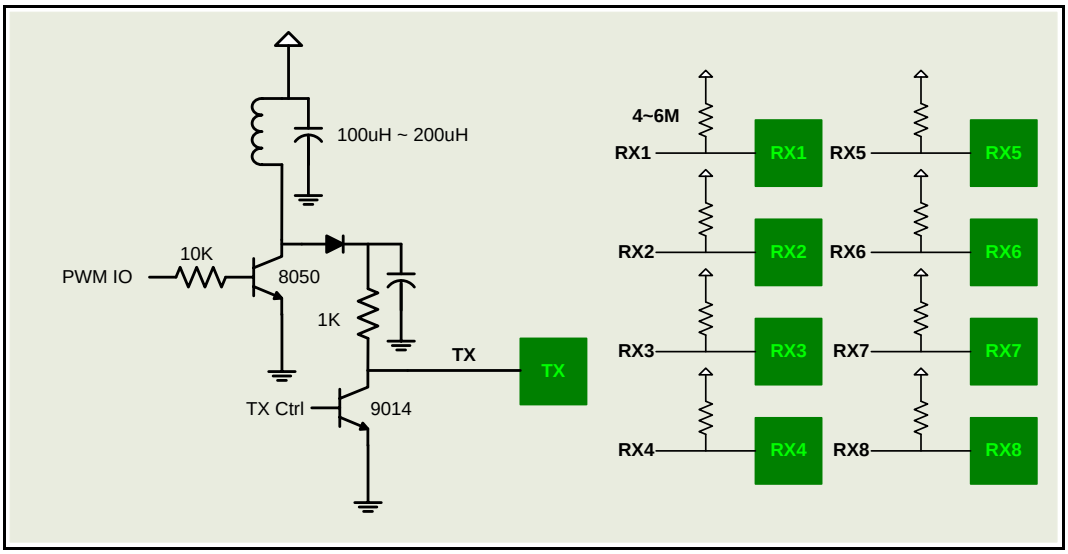

Figure 14

6 PCB Layout Guide

6.1 Definition is as below.

Transmitter pad and its trace will emit short period pulse train to create electric field. It must be more carefully considered and well routed on the PCB. Receiver pads and their traces must take transmitter routing into consideration. Prevent coupling effect while routing on the PCB.

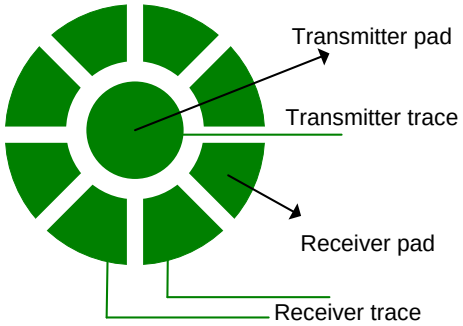

Figure 15

6.2 Cross the transmitter trace and receiver trace as little as possible and only 90° allowed. Please see Figure 16.

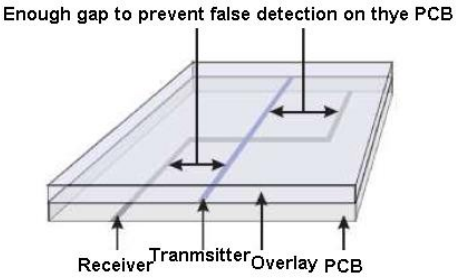

Figure 16

6.3 If transmitter trace and receiver trace must run parallel to each other for a distance, separate them with a ground or leave enough gap between them. The width of ground needed to separate transmitter and receiver trace is greater than 3mm.

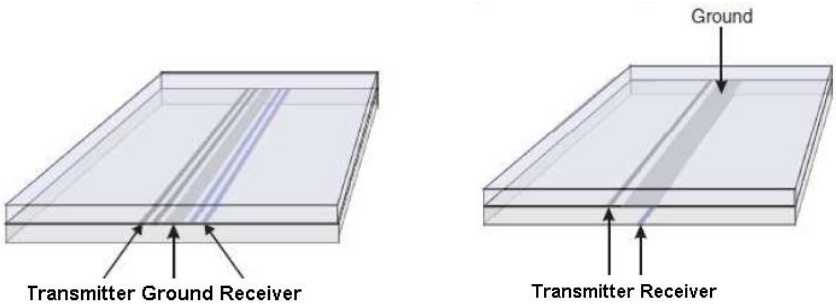

Figure 17

6.4 Avoid routing the transmitter trace beneath a receiver pad or routing the receiver trace beneath a

transmitter pad.

Figure 18

6.5 It is recommended shielding the transmitter trace with ground on both top and bottom of PCB.

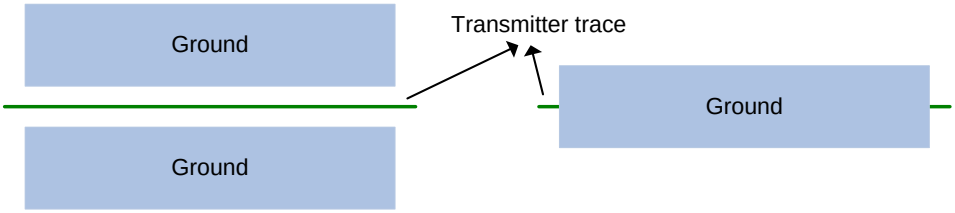

Figure 19

6.6 Single layer PCB is not recommended. If it is a must, leave enough gaps between transmitter traces and receiver traces, and 90° cross is better. If transmitter has to go through two receiver pads, it is recommended routing on the back side of PCB and use ground on the top for shielding. The width of shielding ground is suggested to be greater than 2mm while using boost technology in transmitter.

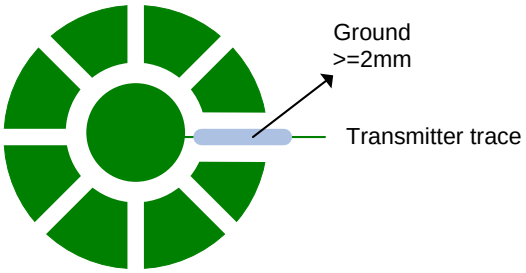

Figure 20

6.7 Prevent the routing shown in following figure.

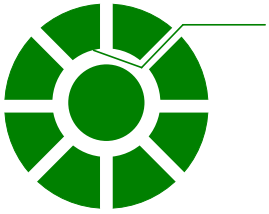

Figure 21

6.8 The gap between transmitter pad and receiver pad should be greater than or equal to the overlay thickness.

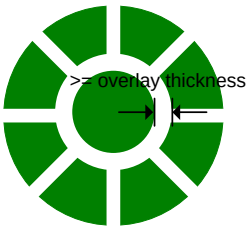

Figure 22

6.9 In the matrix type applications, separate the transmitter traces and receiver traces to prevent false detection.

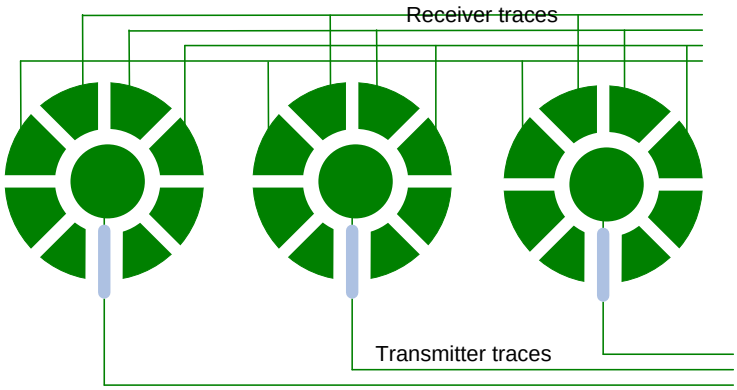

Figure 23

6.10 If high resistivity materials such as carbon or conductive ink are used inside the tag, try to make the trace wider to lower the equivalent impedance.

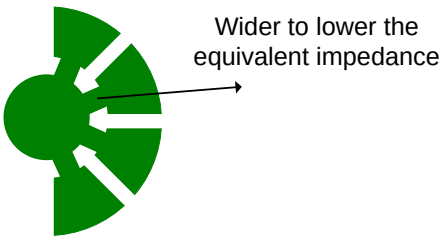

Figure 24