



## 1. Introduction

The purpose of this document is to give a general guide line of the design of a Bluetooth antenna for use with a Bluetooth module without an integrated on-board antenna

## 2. Table of contents

1. Introduction.....	1
2.0 Type of Antennas.....	2
2.1.Printed Circuit Board (PCB) Antenna .....	2
2.1.1.Inverted-F (PIFA) Antenna.....	2
2.1.2. Meandering Line Antenna .....	3
3.0 Reference Designs .....	3
Mounting recommendations .....	5
3.0.1 NEAR Field calculations .....	5
3.1 PCB Manufacturing Specifications.....	6

\*\*\* Important: The document is offered without warranty and ZBA is unable to accept any liability for direct or consequential loss associated with their use. It is therefore important for designers to ensure that their antenna design is properly evaluated in a Design Verification Test. \*\*\*

### 3. Type of Antennas

There are many different types of antennas and relevant design at the 2.4GHz-2.5GHz are:

- Dipole Antennas
- Sleeve Antennas
- Helical Antennas
- Whip (Monopole) Antennas
- Ceramics Chip Antennas
- Slotted Antennas
- Printed/Planar Wiggle Antennas
- Patch Antennas

### 4. Printed Circuit Board (PCB) Antenna

General guide line of two types of printed antennas

Invert-F (PIFA)

Meander Line Antenna

#### 4.1. Inverted-F (PIFA) Antenna

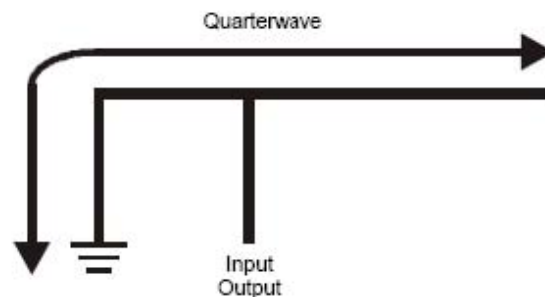


Figure 4.1 Invert-F antenna

The inverted-F is a quarterwave antenna. It is bent into an L-shape. The shorterside is connected to ground point. The longer side is left open circuit at the end of the trace. The feeding point is located somewhere between the ground point and the open end.

On the PCB, this antenna is printed on the top layer and a ground plane is



ZBA, Inc

placed near the antenna on the top layer. There must not be a ground plane underneath the antenna.

#### 4.2. . Meandering Line Antenna



Figure 4.2 Meander Line Antenna

The length of the meander line antenna is difficult to predict. It is usually a bit longer than a quarterwave but dependent on its exact geometry and proximity to the ground plane.

### 5. Reference Designs

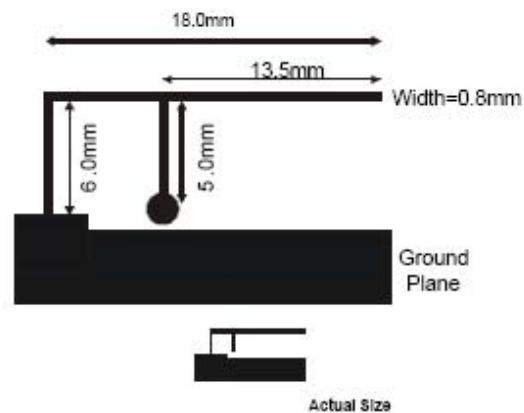
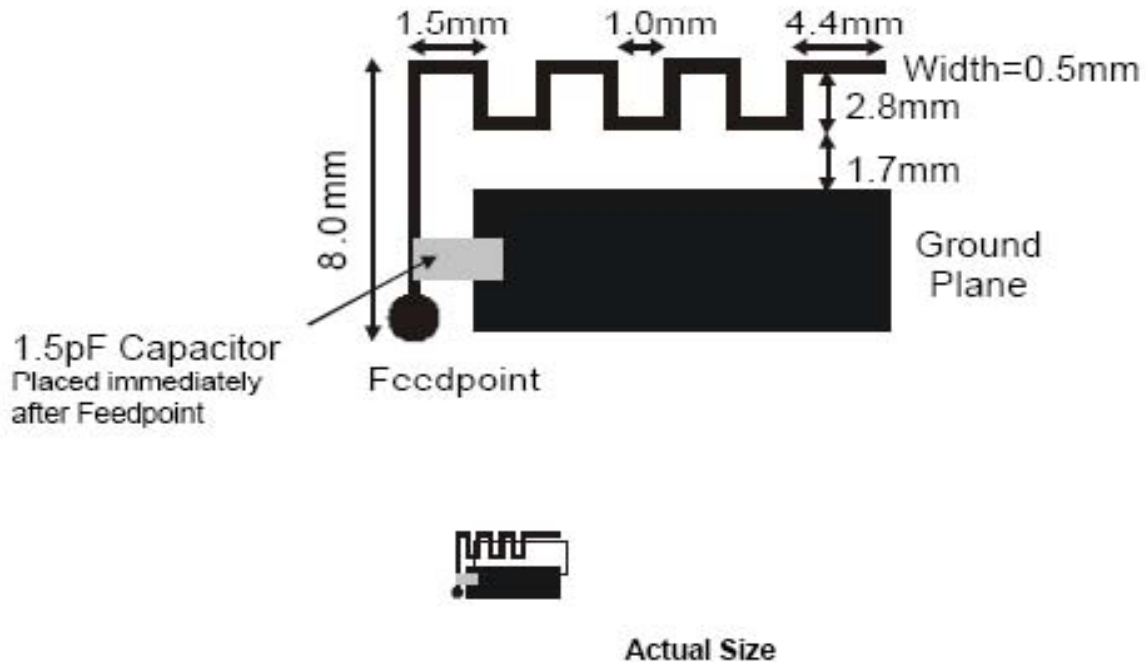




Figure 5.1 Approximate Dimensions of invert-F antenna



Note 1 Two antennas are implemented on a two-layer board using low-cost FR-4 material.

Note 2: ZBA recommends keeping metal object as far away from the antenna as possible. Keeping metallic objects out of the near field is usually adequate. See near field calculation below

NOTE 3: It is strongly recommended that the antenna is placed along one edge of the Printed Circuit board with the antenna placed at the minimum spacing to the edge of the PCB.

NOTE 4: There should be no ground plane or components

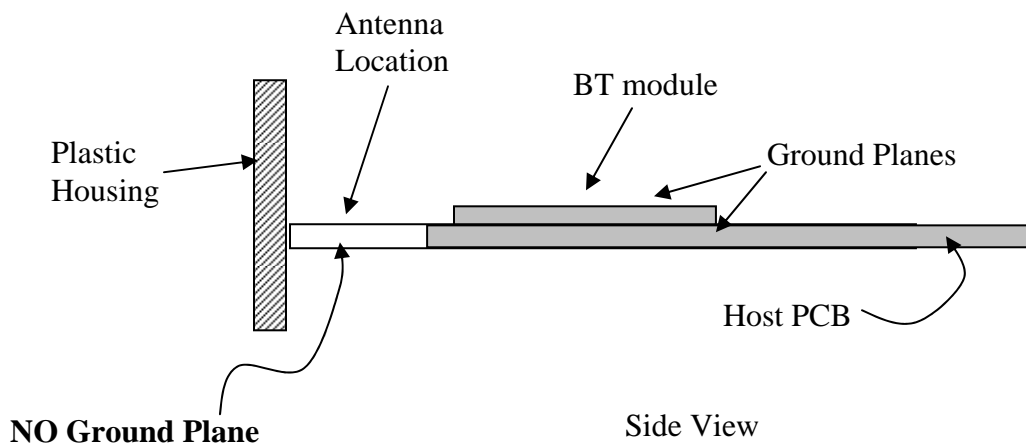


mounted on the underside of the PCB directly under the antenna.

Note 5: It is recommended that components and ground planes on either side of the PCB stay at least 3mm away from any point on the active antenna the horizontal dimension.

## 6. Mounting recommendations

For maximized performance please orient the device with the antenna as close to the outside of the housing as possible. Best performance will occur if the underlying PCB does not have a ground plane under the area where the antenna is located. The nearest ground plane should be at least determined by the above recommendations..



## 7. *NEAR Field calculations*

$$\text{NearField} = 2D^2/\lambda$$

Where D is the largest dimension of the antenna, in this case of these antennas, this is approximately a quarterwave ( $\lambda/4$ ), where  $\lambda$  is the wavelength of the signal in freespace.

At Bluetooth frequency,  $\lambda=122\text{mm}$  in freespace:



Substituting  $D=\lambda/4$  into the Near Field equation gives:

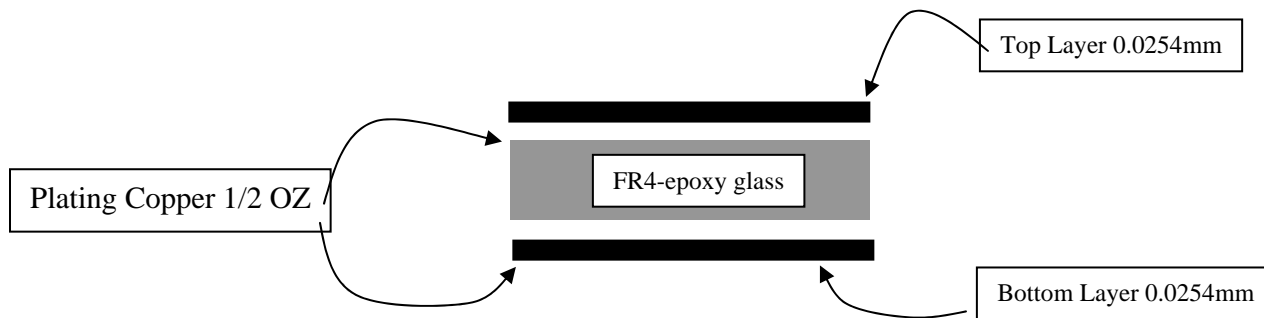
Near Field =  $\lambda/8$ .

Therefore:

Near Field =  $122/8 \text{ mm} = 15.25 \text{ mm}$

## 8. PCB Manufacturing Specifications

This section gives brief specifications for printed circuit board designer to provide PCB manufactures detailed instructions.



### Material

Type FR-4 epoxy glass laminate and prepreg

HTE Copper 1/2 oz copper foil external layer

Overall metal-to-metal thickness  $0.8 \text{ mm} \pm 10\%$  ( $0.031 \text{ inches} \pm 10\%$ )

### Copper Plating

In through-holes  $0.0254 \text{ mm}$  ( $0.001 \text{ inch}$ ) minimum

### Copper Finish

Tin or gold finish ( $10 \text{ } \mu\text{-inch}$  minimum)

### Manufactured boards

To be accordance with performance standard IPC-6011/6012 class 2 board