

# Proper PCB Design for Embedded Antennas

## Application Note AN-00502



### Introduction

Embedded antennas are ideal for products that cannot use an external antenna. The reasons for this can range from ergonomic or aesthetic reasons or perhaps the product needs to be sealed because it is to be used in a rough environment. It could also simply be based on cost. Whatever the reason, embedded antennas are a popular solution.

Embedded antennas are generally quarter wave monopole antennas. These are only half of the antenna structure with the other half being a ground plane on the product's circuit board. The details of antenna operation are beyond the scope of this application note, but suffice it to say that the layout of the PCB becomes critical to the RF performance of the product.

### Common PCB Layout Guidelines

While each antenna style has specific requirements, there are some requirements that are common to all of them.

- Use a manufactured board for testing the antenna performance. RF is very picky and perf boards or other “hacked” boards will at best give a poor indication of the antenna’s true performance and at worst will simply not work. Each antenna has a recommended layout that should be followed as closely as possible to get the performance indicated in the antenna’s data sheet. Most Linix antennas have an evaluation kit that includes a test board, so please contact us for details.
- The antenna should, as much as reasonably possible, be isolated from other components on your PCB, especially high-frequency circuitry such as crystal oscillators, switching power supplies, and high-speed bus lines. Everything in the antenna’s near field (within one wavelength) has an impact on the radiation pattern and performance, so the antenna needs to be on its own. This is contrary to much of today’s designs where everything is compacted as much as possible, but it is what is necessary to get the most out of the antenna.
- Keep traces away from the antenna. Traces can become antennas themselves, frequently at a harmonic of the operational frequency. This can cause issues when going for regulatory certifications. This includes traces under the antenna itself on any layer of the board.
- The antenna is only half of the complete antenna structure. The other half is a ground plane on the circuit board. The dimensions of the plane vary by antenna and are described later in this note. However,



Figure 1: Linix Embedded Antennas

the plane should be as solid as possible. It can act as the ground connection for other circuits on the board. More details on the ground plane can be found in application notes AN-00500 and AN-00501.

- The antennas are tuned to be 50 ohms at the frequency of operation. The connection to the radio needs to be a single-ended 50-ohm transmission line. This is typically either a microstrip line or a co-planar waveguide. Appendix 1 goes into these types of transmission lines in more detail.
- The product's enclosure needs to be non-conductive. Embedded antennas cannot be used effectively in metal, carbon fiber or some fiberglass enclosures. Placing non-conductive panels in metal enclosures does allow some signal to get out in the direction of the panel, but has a significant impact on the overall performance. Not only does it potentially reduce the radiated power, it also focuses the power just like a flashlight focuses the light from a bulb. This gives great range during initial testing, but can be a great disappointment when regulatory compliance testing requires the transmitter power to be reduced. This could result in much less range than was achieved during initial testing and greatly affect the product's performance.
- Just as traces on the PCB need to be kept away from the antenna, unshielded wires and wire harnesses inside the enclosure also need to be kept away for the same reason. Be sure to secure them so that they will not come loose and fall across the antenna.

Additional PCB layout and product design requirements are discussed for each of the Linx embedded antennas in the following sections. Each of these antenna families are tested on a fixture with a specific layout. All of the published specifications, Voltage Standing Wave Ratio (VSWR) plots and radiation patterns are obtained on these fixtures. The closer these layouts are replicated in the final product, the closer the product's performance will be to the published specifications. However, since the antenna's final performance is critically dependent on the product's layout and construction, it will likely differ some from published materials once incorporated into the end product.

## CHP Series



The CHP Series ceramic chip antennas are available in three frequencies; 868MHz, 916MHz and 2.4GHz. The 868 and 916MHz versions are tested on one test fixture and the 2.4GHz is tested on another, smaller test fixture. These fixtures are shown in the figures below.

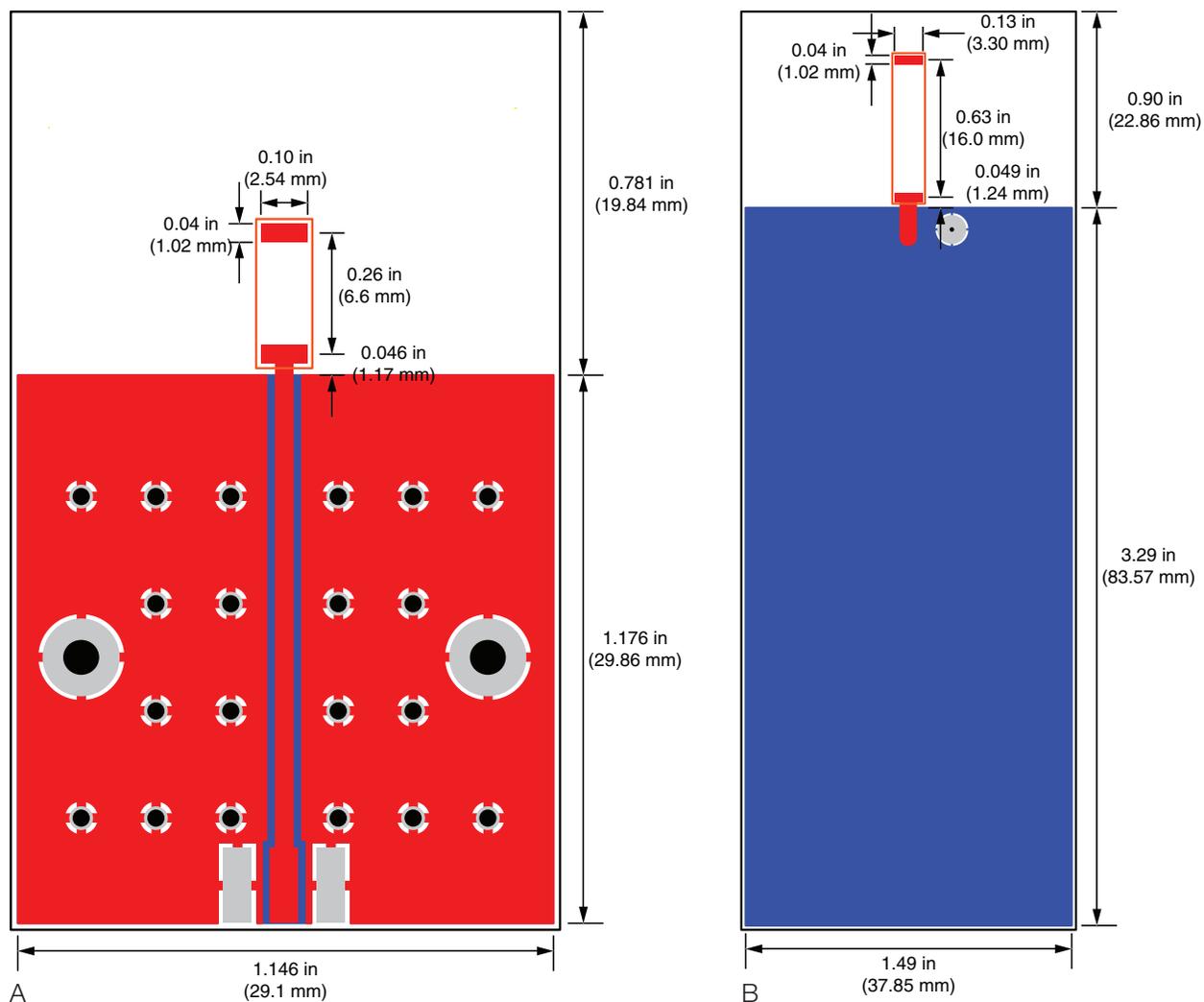


Figure 2: The CHP Series Test Fixtures; A) 2.4GHz; B) 868 and 916MHz

**Top layer copper is red, bottom layer is blue.** There are a few critical features to notice on these boards.

- There are no traces, planes or any copper under the antenna or to its sides on any layer of the board. Anything conductive in this area will impact the radiation pattern and the antenna's performance. There is only one pad in this area and it is for the antenna itself. This pad is for physical support and should not be electrically connected.
- There is a solid ground plane on the bottom layer. The size of this ground plane is important. Anything smaller will shift the antenna's resonant frequency higher and could push it out of the band if it is too small. Anything larger will shift the antenna's frequency lower, but the amount of the shift will not be as much as what is caused by a smaller ground plane. So it is better to have too much plane than not enough.

- The ground plane can be the common ground for the radio and other circuits. These circuits can be placed on the top layer above the plane.
- The feed line to the antenna from the radio is a 50-ohm transmission line. The antenna is matched to 50 ohms, so a radio or radio module that is already matched should not need any other components. Just about all RFIC manufacturers have a recommended circuit to match their radio chips to 50 ohms, so follow their guidelines and no other components should be required.
- The 2.4GHz test board has ground plane on the top and bottom layers. This allows for the implementation of a Co-Planar Waveguide (CPWG) for the feed to the antenna. A ground plane on the bottom layer and a microstrip feed are used on the 868 and 916MHz test board. Both methods are acceptable and either can be used for any frequency. The CPWG was used on the 2.4GHz board to give a smaller trace that fit better into the SMA connector pads at the bottom of the board. Any transmission line can be used as long as it is a controlled 50-ohm impedance.

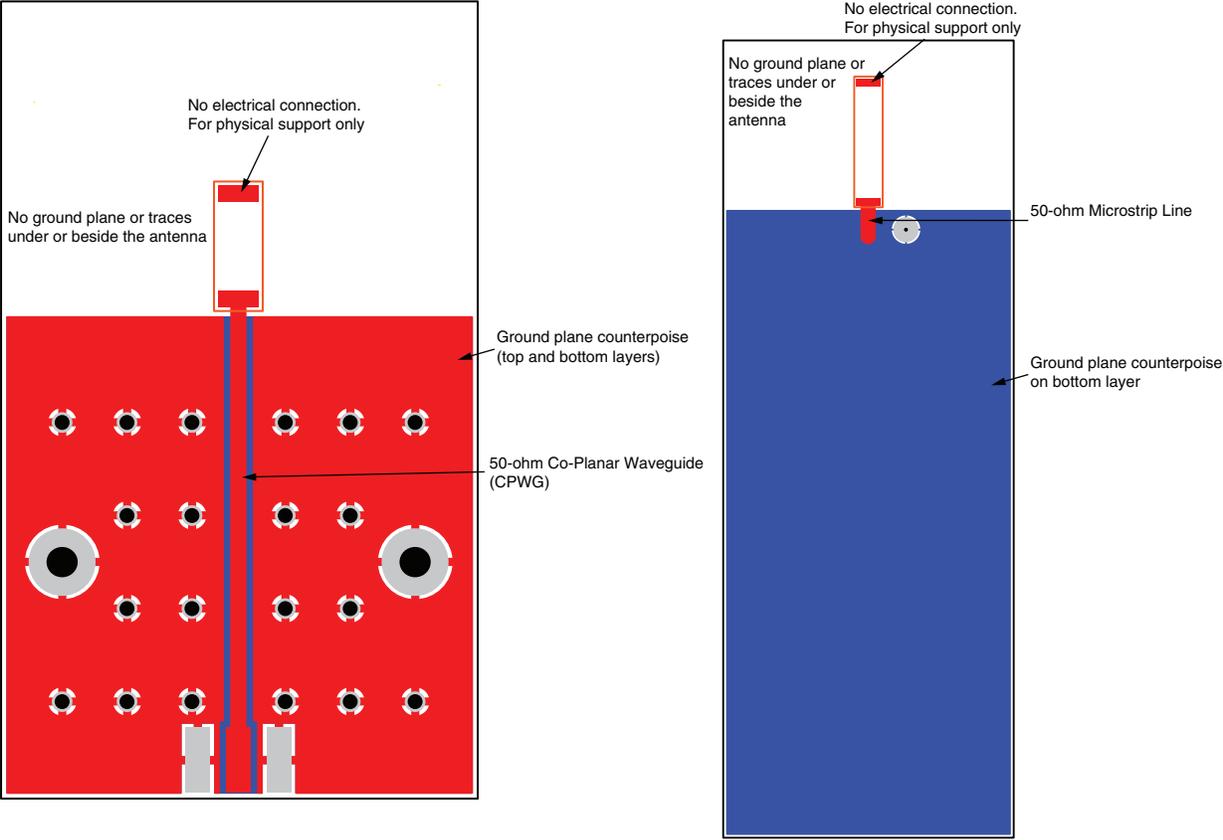


Figure 3: The CHP Series Test Fixtures

## JJB Series



The JJB Series comes in two varieties; straight and right angle. Each variety has three frequencies (868MHz, 916MHz and 2.45GHz) and its own test fixture that is used for all of the available frequencies. These fixtures are shown in the figures below.

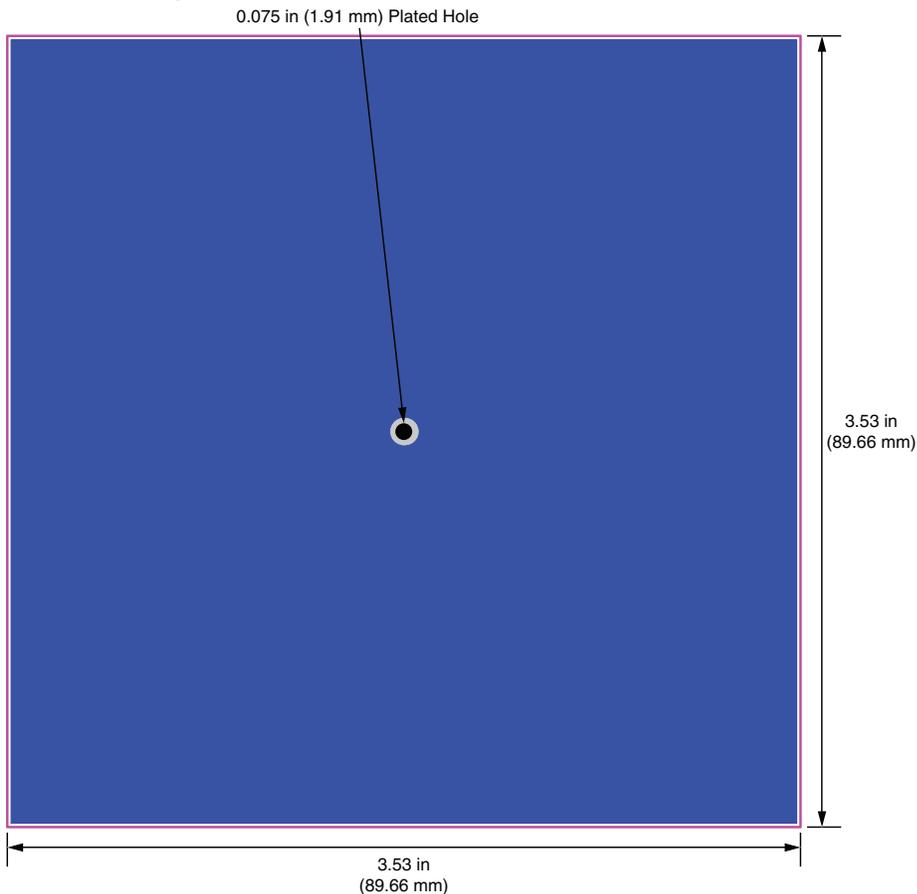


Figure 4: The JJB-ST Series Test Fixture

- The straight version is in the center of a 3.5 x 3.5 inch (90 x 90mm) ground plane. The plane is on the bottom layer and its size is important. Anything smaller will shift the antenna's resonant frequency higher and could push it out of the band if it is too small. Anything larger will shift the antenna's frequency lower, but the amount of the shift will not be as much as what is caused by a smaller ground plane. So it is better to have too much plane than not enough.

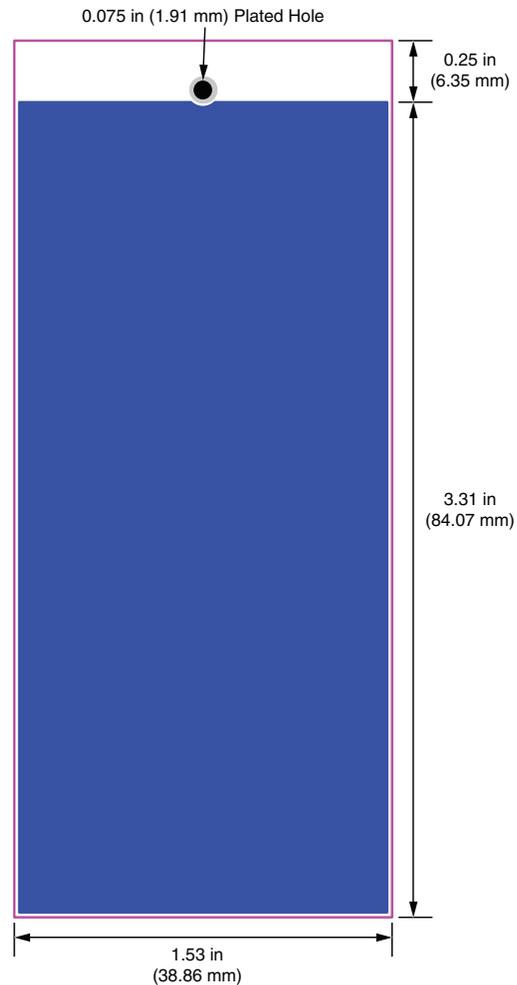


Figure 5: The JJB-RA Series Test Fixture

- The right angle version is used with a 1.5 x 3.3 inch (38.1 x 83.82mm) ground plane. Like the straight version, the plane is on the bottom layer and its size is important. There are no traces, planes or any copper beneath or to the sides of the head of the antenna. Anything conductive in this area impacts the radiation pattern and the antenna's performance. The head of the antenna can hang off the board, but it is better to have the board there for physical support. Without the board there, mechanical stress, such as dropping the board, could cause the antenna's solder joint to break. The extra board space is not required and is somewhat application specific, but it is recommended.
- The ground plane can be the common ground for the radio and other circuits. These circuits can be placed on the top layer above the plane.
- The feed line to the antenna from the radio is a 50-ohm transmission line. The antenna is matched to 50 ohms, so a radio or radio module that is already matched should not need any other components. Just about all RFIC manufacturers have a recommended circuit to match their radio chips to 50 ohms, so follow their guidelines and no other components should be required.

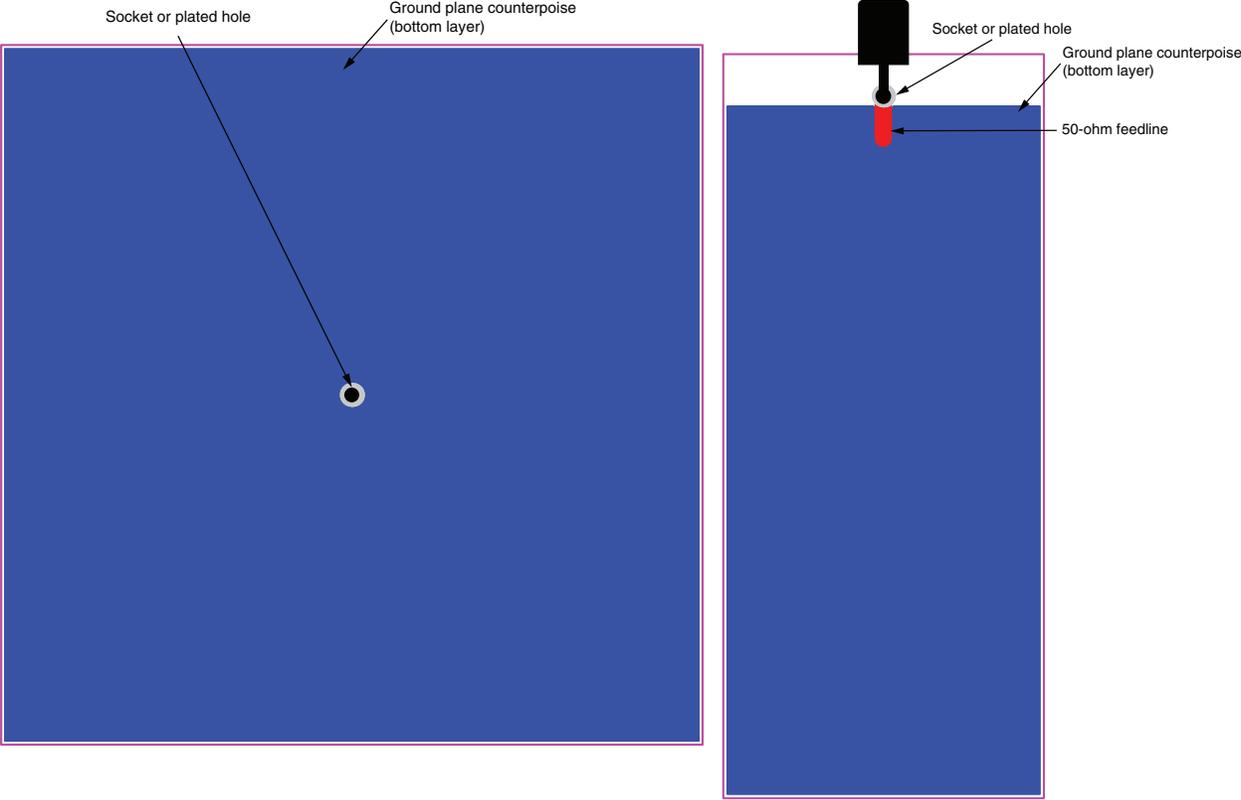


Figure 6: The JJB Series Test Fixtures

## SP Series



All frequencies of the SP Series are the same size and use the same test fixture, which is shown in Figure 7.

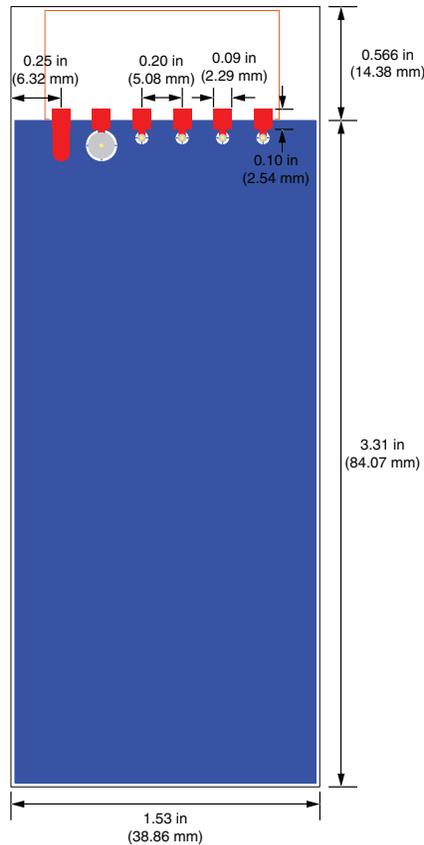


Figure 7: The SP Series Test Fixture

- There are no traces, planes or any copper under the antenna or to its sides on any layer of the board. Anything conductive in this area will impact the radiation pattern and the antenna's performance.
- The ground plane starts in the middle of the antenna's pads and extends away from the antenna. The size of this ground plane is important. Anything smaller will shift the antenna's resonant frequency higher and could push it out of the band if it is too small. Anything larger will shift the antenna's frequency lower, but the amount of the shift will not be as much as what is caused by a smaller ground plane. So it is better to have too much plane than not enough.
- The ground plane can be the common ground for the radio and other circuits. These circuits can be placed on the top layer above the plane.

- The feed line to the antenna from the radio is a 50-ohm transmission line. The antenna is matched to 50 ohms, so a radio or radio module that is already matched should not need any other components. Just about all RFIC manufacturers have a recommended circuit to match their radio chips to 50 ohms, so follow their guidelines and no other components should be required.

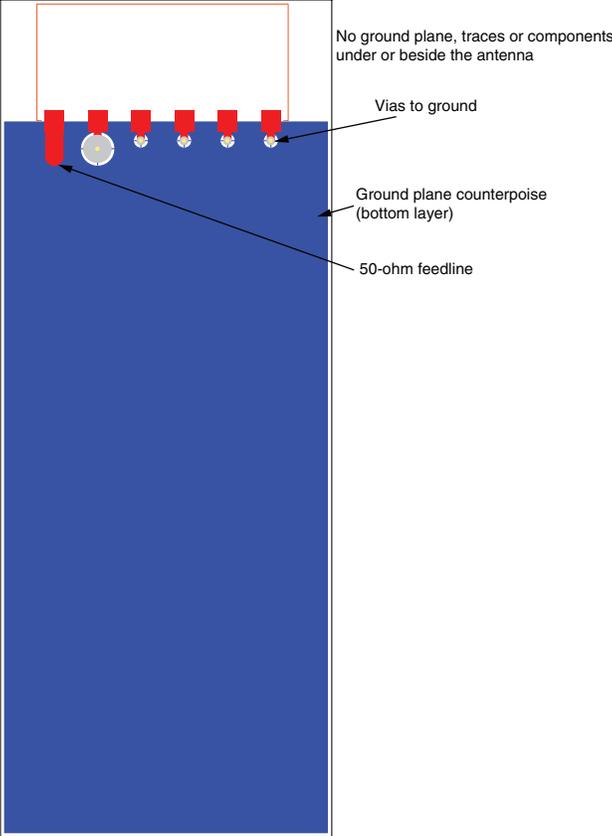


Figure 8: The SP Series Test Fixture

## uSP Series



All frequencies of the uSP Series are the same size, but there are two test fixtures. The 2.45GHz version is tested on one and the rest of the frequencies are tested on the larger fixture. These are shown in Figure 9.

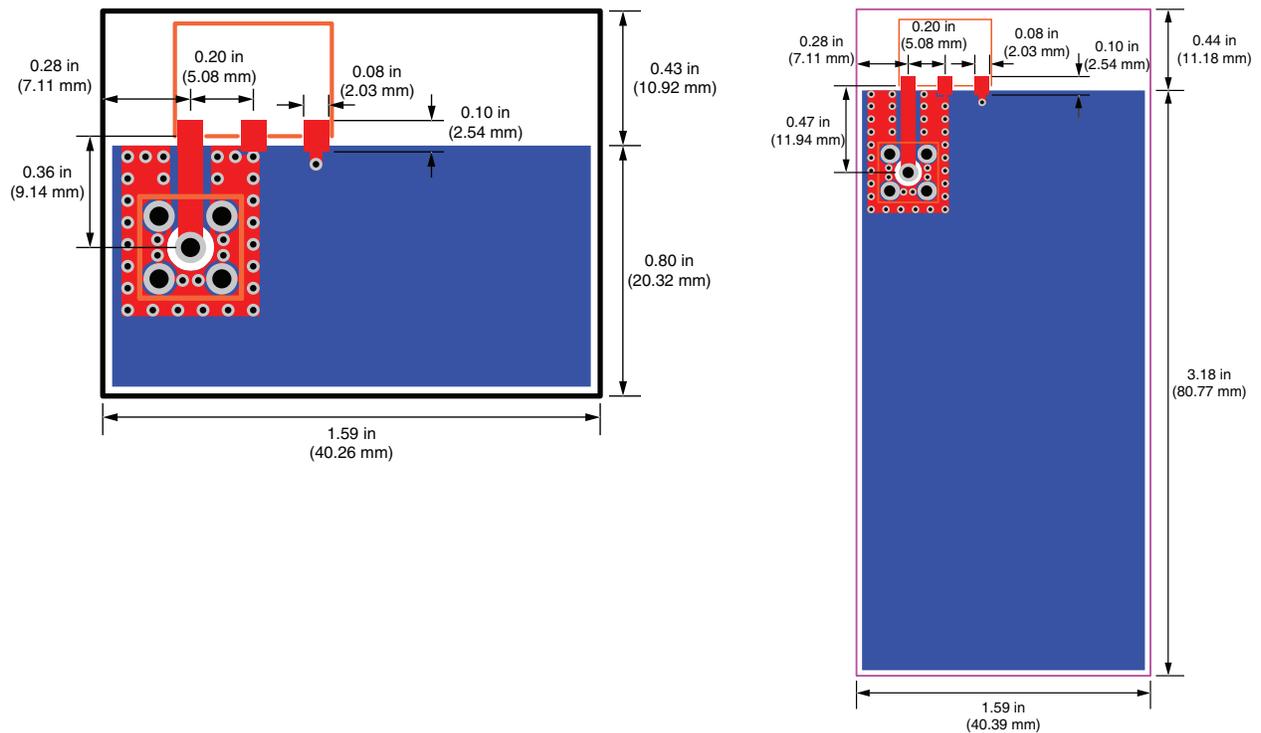


Figure 9: The uSP Series Test Fixtures

- There are no traces, planes or any copper under the antenna or to its sides on any layer of the board. Anything conductive in this area will impact the radiation pattern and the antenna's performance.
- There is a solid ground plane on the bottom layer. The size of this ground plane is important. Anything smaller will shift the antenna's resonant frequency higher and could push it out of the band if it is too small. Anything larger will shift the antenna's frequency lower, but the amount of the shift will not be as much as what is caused by a smaller ground plane. So it is better to have too much plane than not enough.
- The ground plane can be the common ground for the radio and other circuits. These circuits can be placed on the top layer above the plane.

- The feed line to the antenna from the radio is a 50-ohm transmission line. The antenna is matched to 50 ohms, so a radio or radio module that is already matched should not need any other components. Just about all RFIC manufacturers have a recommended circuit to match their radio chips to 50 ohms, so follow their guidelines and no other components should be required.
- The test fixtures have ground plane on the top and bottom layers. This allows for the implementation of a Co-Planar Waveguide (CPWG) for the feed to the antenna. The CPWG was used to give a smaller trace that fit better into the SMA connector pads. Any transmission line can be used as long as it is a controlled 50-ohm impedance.
- It is recommended to keep at least 0.21" (5.33mm) from the antenna to any components or traces. This keeps object's out of the antenna's near field and reduces the impact on the radiation pattern. Ideally, there will not be anything within 1 wavelength of the antenna, but this is not really practical. This clearance has shown to be a reasonable compromise.

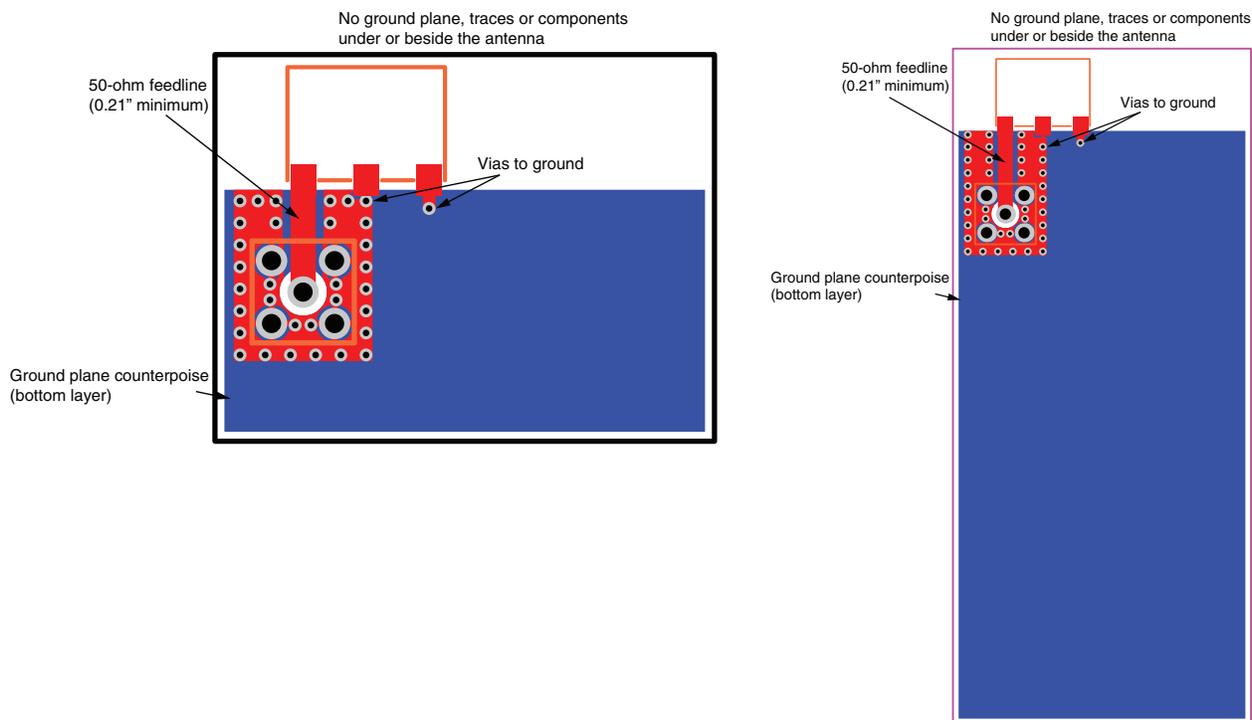
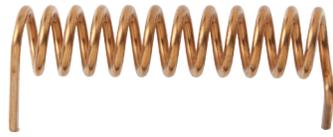


Figure 10: The uSP Series Test Fixtures

## HE Series



The HE Series has several frequencies and two sizes. These are tested on different fixtures based on frequency. The 315MHz version is tested on the largest fixture. This is shown in Figure 11.

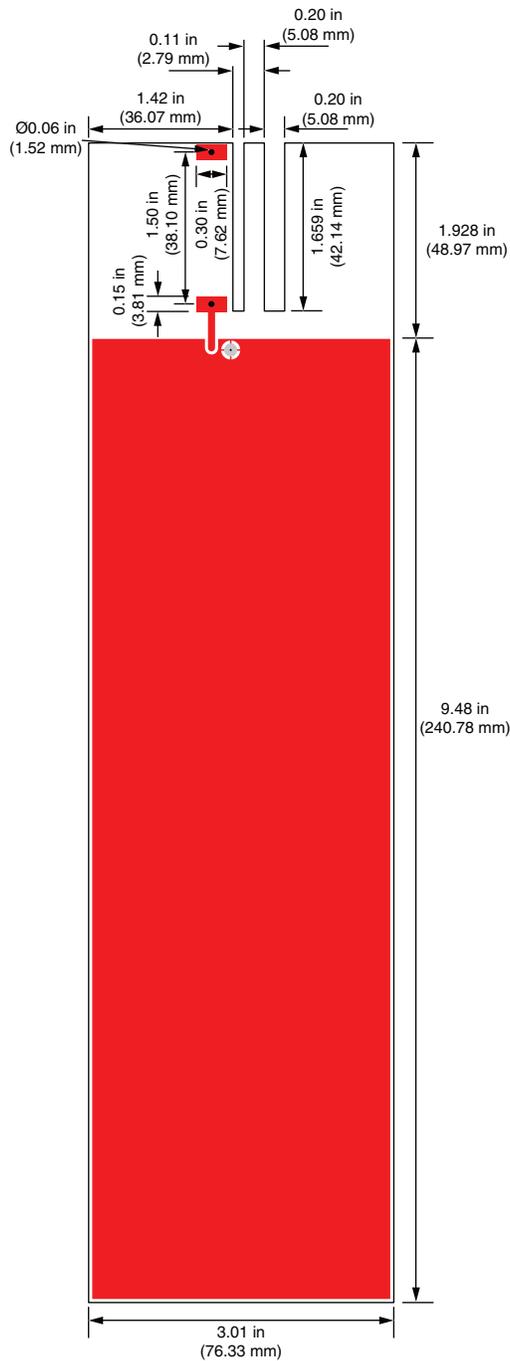


Figure 11: The 315MHz HE Series Test Fixture

The 418MHz and 433MHz versions are tested on a smaller fixture.

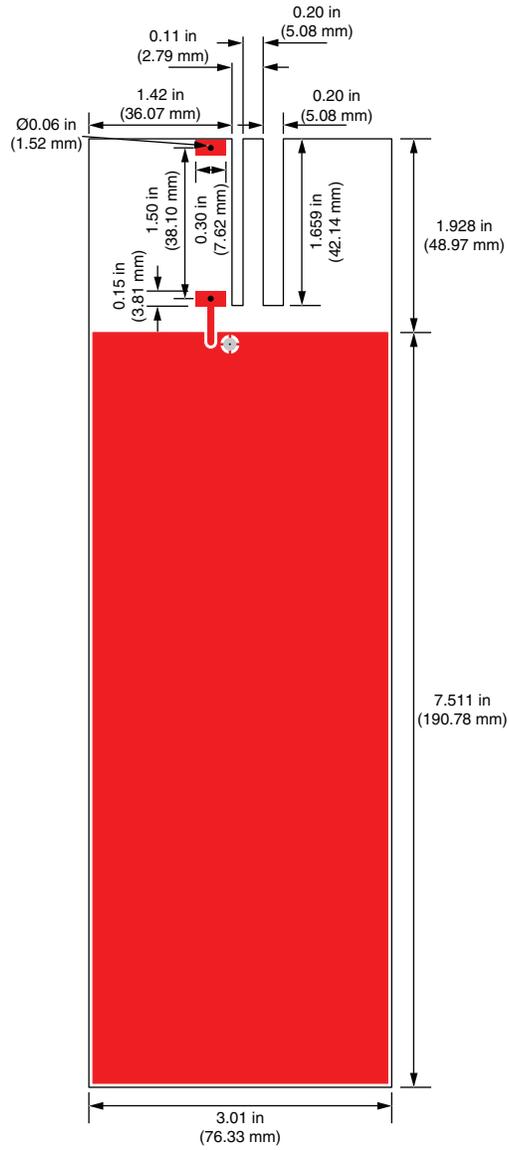


Figure 12: The 418 and 433MHz HE Series Test Fixture



- The ground plane can be the common ground for the radio and other circuits. These circuits can be placed on the top layer above the plane.
- The feed line to the antenna from the radio is a 50-ohm transmission line. The antenna is matched to 50 ohms, so a radio or radio module that is already matched should not need any other components. Just about all RFIC manufacturers have a recommended circuit to match their radio chips to 50 ohms, so follow their guidelines and no other components should be required.
- Some of the feed line is not above the ground plane on the lower frequency fixtures. This adds to the antenna's length and adjusts its resonant frequency. The antenna is designed for this extra length so it should be incorporated into the board layout.

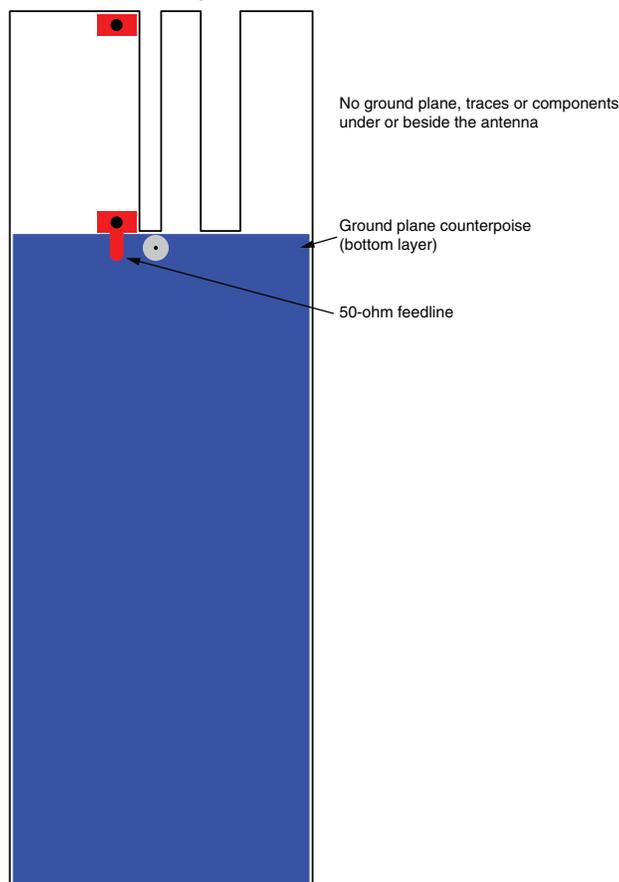


Figure 14: The 916MHz HE Series Test Fixture

## Tricks of the Trade

In some designs it is not possible or practical to include as much ground plane as is necessary. In other cases the packaging, wires or other components of the design make the plane much larger than is recommended. These situations can cause the antenna to resonate out of band. There are some ways to dial the antennas back in band.

The first way is to incorporate a PI network between the antenna and the radio. This is two capacitors to ground on either side of a series inductor. The values can be selected to electrically tune the antenna. It does take test equipment such as a network analyzer to get this right.

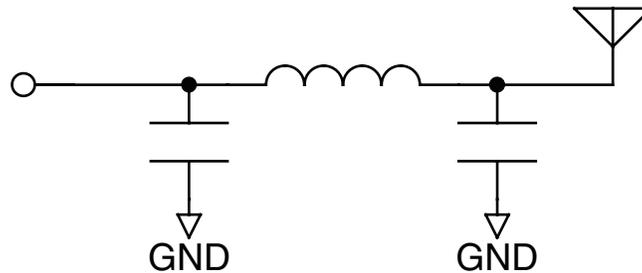


Figure 15: PI Network

The PI network is good for shifting an antenna a few MHz. Anything larger and it will be difficult to pull the antenna in band and have good efficiency. If the antenna is pulled higher in frequency, then the antenna trace can be made longer to increase the length of the antenna. This additional trace would not be a microstrip or CPWG. There should be no ground plane under or around the trace. This length can be adjusted to dial the antenna into the correct band.

If the resonant frequency is lower, then it may be possible to get an antenna at a higher frequency and have the board pull it lower in band. This depends on how much the antenna is pulled and what frequencies are available in the antenna of choice.

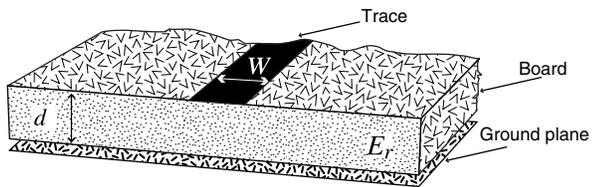
If all else fails, it may be necessary to investigate a custom antenna design. Please contact Linx for details on this.

## Appendix 1, Transmission Lines for Embedded Antennas

A transmission line is a medium whereby RF energy is transferred from one place to another with minimal loss. It is designed for a specific characteristic impedance to match the antenna to the radio. This is a critical factor, especially in high-frequency products because the trace leading to the antenna can effectively contribute to the length of the antenna, changing its resonant frequency. This detuning increases energy loss in the system and reduces overall range of the radio link. In order to minimize loss and detuning, some form of transmission line between the antenna and the radio should be used. There are several kinds of transmission lines but two are commonly used for low-cost embedded radios; a microstrip line and a co-planar waveguide.

### Microstrip Transmission Lines

A microstrip is a PCB trace that runs over a ground plane. There are several factors that contribute to its characteristic impedance, but the two most critical ones are distance from the ground plane and width of the trace. The calculations for this are shown in Figure 16 and Figure 17 shows some example calculations.



$$E_e = \frac{E_r + 1}{2} + \frac{E_r - 1}{2} \cdot \frac{1}{\sqrt{1 + 12d/W}}$$

$$Z_0 = \begin{cases} \frac{60}{\sqrt{E_e}} \cdot \ln\left(\frac{8d}{W} + \frac{W}{4d}\right) & \text{For } \frac{W}{d} \leq 1 \\ \frac{120\pi}{\sqrt{E_e} \cdot \left(\frac{W}{d} + 1.393 + 0.667 \cdot \ln\left(\frac{W}{d} + 1.444\right)\right)} & \text{For } \frac{W}{d} \geq 1 \end{cases}$$

$E_r$  = Dielectric constant of PCB material

Figure 16: Microstrip Formulas

Example Microstrip Calculations			
Dielectric Constant	Width / Height Ratio (W / d)	Effective Dielectric Constant	Characteristic Impedance ( $\Omega$ )
4.80	1.8	3.59	50.0
4.00	2.0	3.07	51.0
2.55	3.0	2.12	48.8

Figure 17: Example Microstrip Calculations

## Grounded Coplanar Waveguide

A grounded coplanar waveguide is a PCB trace that has ground plane on both sides and on a lower layer.

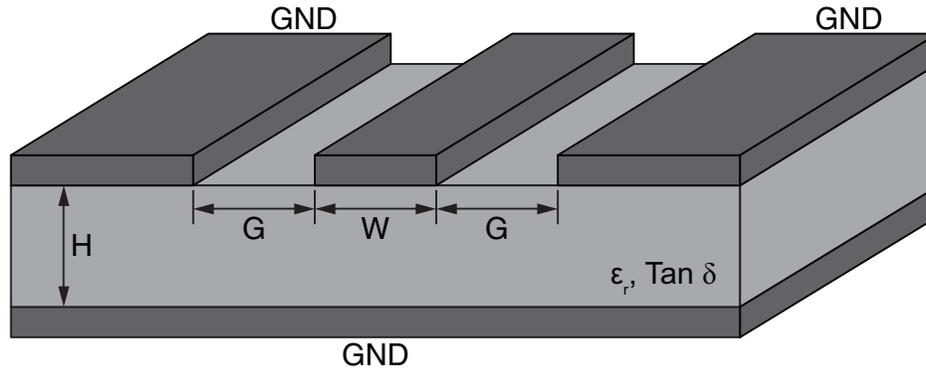


Figure 18: Grounded Coplanar Waveguide Construction

This structure allows more precise control over the line impedance and results in smaller trace widths at a given impedance. The calculations for the design get fairly complicated, so online calculators are generally used or calculators built into design software.

Because the ground plane is on both sides of the trace and on the bottom, it is important to ensure that the planes are at the same impedance. Vias are typically added in rows along the edge of the gaps to connect the top plane to the bottom plane. This is referred to as “fencing” or “stapling”. When adding vias, the rule-of-thumb is to add them at spacings of  $1/8$  of a wavelength or less. This gives good isolation and makes the ground plane look solid.

For the best isolation, the rule is to space vias at  $1/20$  of a wavelength or less. However, this is generally overkill for most commercial products and the  $1/8$  rule is more practical.