# Simulation and Measurement of an Inverted-F Antenna Printed on a Wireless Sensor Networks PCB

SI Team
Tessolve Services Pvt Ltd
www.tessolve.com

# Simulation and Measurement of an Inverted-F Antenna Printed on a Wireless Sensor Networks PCB

## Introduction:

The use of 2.4 GHz industrial, scientific and medical (ISM) band is becoming an important means of wireless communication. Wireless local area networks (WLAN), wireless internet at any access-point equipped building, Bluetooth and Zigbee wireless networks all utilize the 2.4 GHz ISM Band. Therefore, the development of appropriate antenna design is imperative.

A group of candidate antenna structures that might be considered for use in short-range radio devices is made up of the printed dipole, the microstrip patch and the variants of monopole. The monopole is a variant of the dipole that uses the device's ground plane to provide the other half of the antenna. The Inverted-F Antenna (IFA) falls into this category and is well known for its ability to provide flexibility in impedance matching. The IFA printed on a WSN (Wireless Sensor Networks) PCB incorporates desirable characteristics such as no additional cost, ease of manufacture, compact size, acceptable bandwidth, high efficiency and a good Omni-directional radiation pattern.

In this document, the four layer WSN board is simulated, to see if the IFA has got a return loss below -10dB. The simulated results showed that the return loss is around -6dB at 2.4GHz. The simulated results are compared with measurements on a fabricated board measured using Credence ASL 3000 RF tester. The optimization of the IFA is then performed by altering the inner layers and the optimum distance, to recess the planes has been found through IE3D simulation.

Also, the two layer reference board used in Freescale's SRB (Sensor Reference Board) has been reconstructed in MGRID and simulated to see if simulation matches the lab measurement given by them (Figure 12). The four layer sample board was then drawn as two layer boards and simulated to see if the performance increased in the two layer board

## Basic Antenna Theory:

Every structure carrying RF current generates an electromagnetic field and can radiate RF power to some extent. Likewise, an external RF field can introduce currents in the structure. This means that theoretically, any metallic structure can be used as an antenna. However, some structures are more efficient in radiating and receiving RF power than others. Simply, the antenna is an impedance matching device between PCB (50 Ω) and air (377 Ω). The good antenna structure should be a reasonable size compared to the wavelength of the RF field. A natural size is a half wavelength. A half wavelength corresponds to approximately 6 cm (in air) in the 2.4 GHz ISM band for dipole antenna. This size is effective because when it is fed with RF power at the center point, the structure is resonant at the half wave frequency. Reducing the size below the natural resonant length can cause low efficiency. A lot of external factors affect antenna's impedance matching, tuning, gain and radiation patterns. An antenna tuned for one set of environmental factors (PCB dimension, different casing and nearby ground connected objects) may not perform at all if put into a new environment, and may require a lot of tuning to achieve even reasonable performance.

**Inverted-F Antenna Geometry and Dimensions:**

Two IFAs are printed on the WSN PCB's both top and bottom layers. The geometry and dimensions (in mm) of the printed IFA from Freescale used in the board are depicted below:

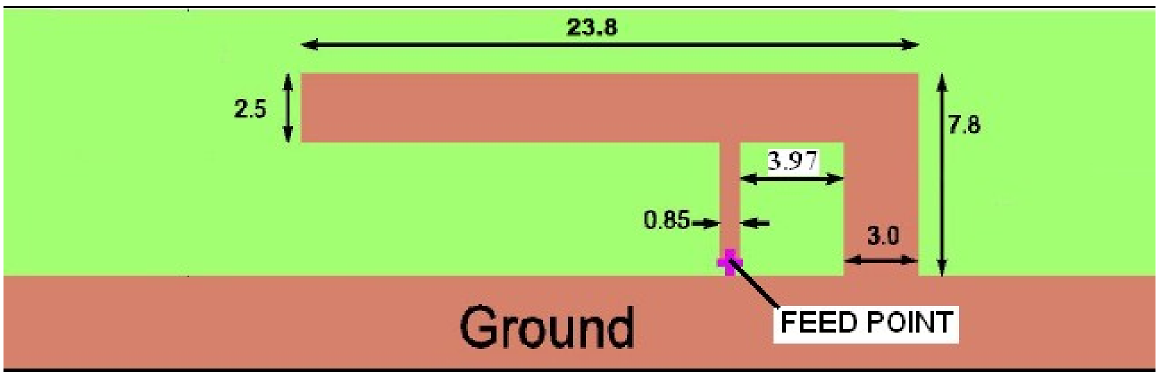


**Figure 1: Printed IFA Geometry**

**WSN System:**

The WSN system to be analyzed is a Freescale's ZigBee (IEEE 802.15.4) device MC13213 mounted on four layers PCB. The approximate board diameter is about 80mm. The board's final thickness excluding soldermask is around 1.042mm. The base laminate material used is FR4 with a dielectric constant of 4.29 and dielectric loss tangent of 0.023. The stackup followed in the WSN PCB is shown below:

| Layer # | Layer Name | Material Description | Thickness (mm) |
|---|---|---|---|
| 1 | **Top** | Copper | 0.048 |
| | Prepreg | FR4 | 0.2 |
| 2 | **Gnd** | Copper | 0.018 |
| | Core | FR4 | 0.51 |
| 3 | **Vcc** | Copper | 0.018 |
| | Prepreg | FR4 | 0.2 |
| 4 | **Bottom** | Copper | 0.048 |
| | | **Total Thickness** | 1.042 |

**Table 1: Stack-up of the WSN PCB**

The printed IFA is placed close to the edge of the board on both the top and bottom layers. The antenna's 3mm width arm is connected to the ground plane on the top and bottom layers. The antennas are connected through vias and RF signal is fed to the antenna on top layer. Excitation of currents in the printed IFA causes excitation of currents in the adjacent ground plane and the resulting electromagnetic field is formed by the interaction of the IFA and an image of itself in the adjacent ground plane.

**Simulation of printed IFA on WSN PCB using IE3D:**

The four layer board along with the IFA is either constructed in the MGrid Layout Editor with the actual stackup defined as shown in Table 1 or imported into MGrid from any one of the various geometry file types (.dxf, .pho) and the appropriate thickness and stackup are defined to replicate the 2-D structure in 3-D environment. Zeland IE3D is accurate and efficient, because, it provides various port schemes for different types of structures and the appropriate port scheme is chosen for the simulation of IFA. The port is set at the feed point location of the antenna for S11 (Return Loss) generation and a small rectangular cut is made in the top layer ground plane for the ease of setting extension ports. The four layer board with IFA imported into MGrid Editor with appropriate dimensions and thickness is shown below:

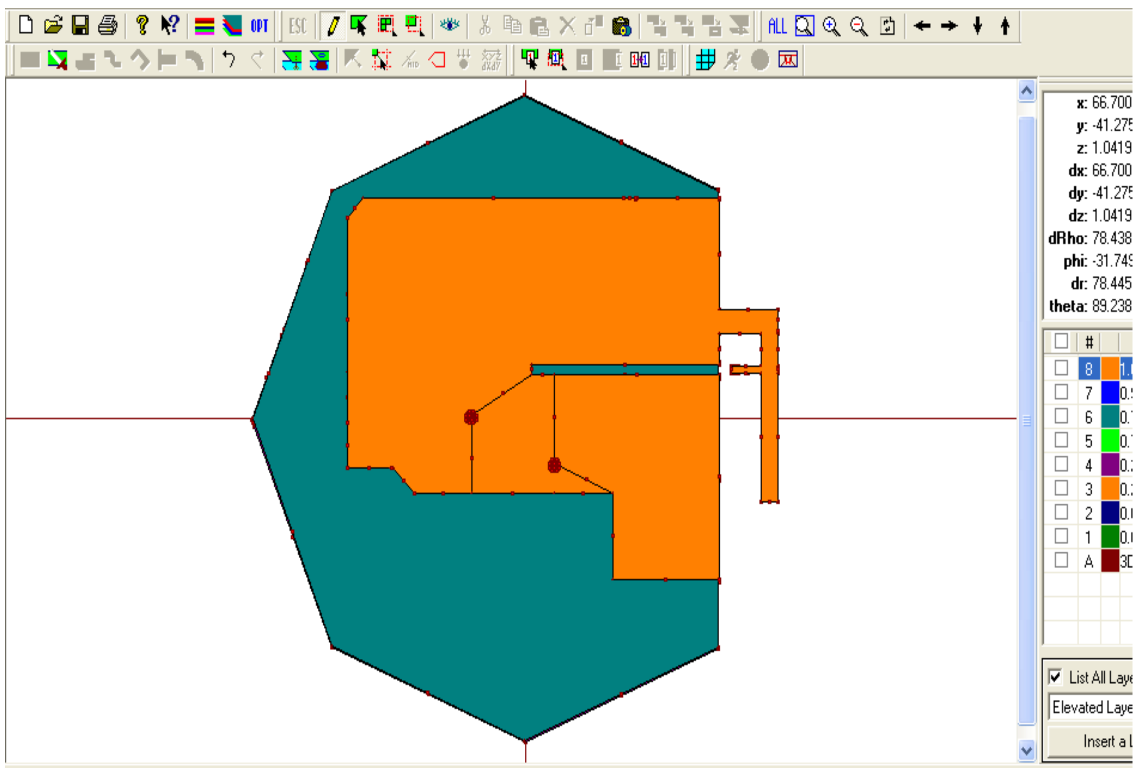


**Figure 2: The board in MGrid Layout Editor**

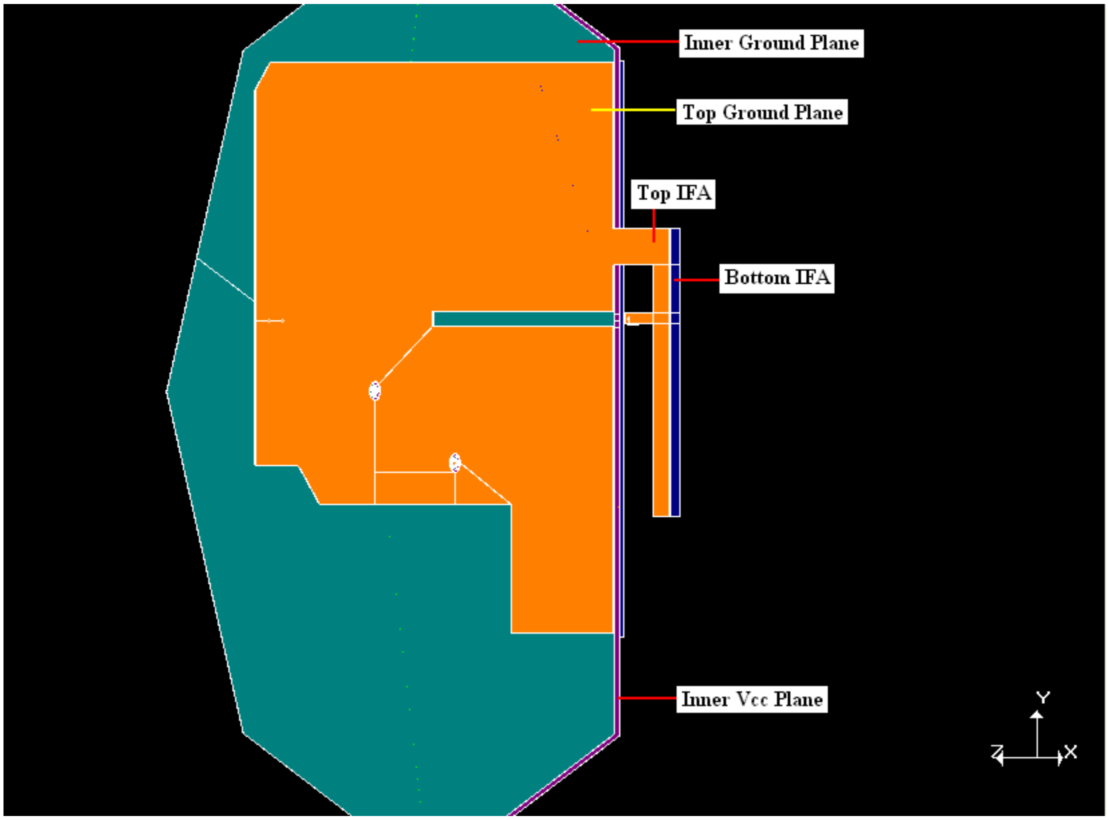


**Figure 3: The board in 3-D View**

The structure is simulated using IE3D Engine's Adaptive Symmetric Matrix solver, with a Meshing frequency of 20 GHz and frequency range from 2 to 3 GHz. The result simulated with 11 frequency points at a step frequency of 100 MHz is given below and the E-Plane Radiation Pattern of the Actual Board is shown in Figure 5:

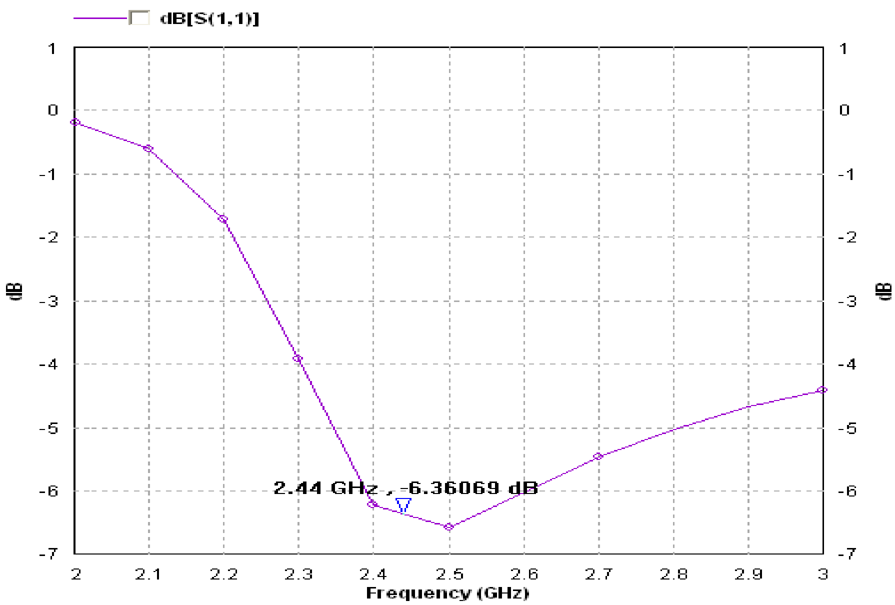


**Figure 4: Return Loss Simulation output of the actual Board**

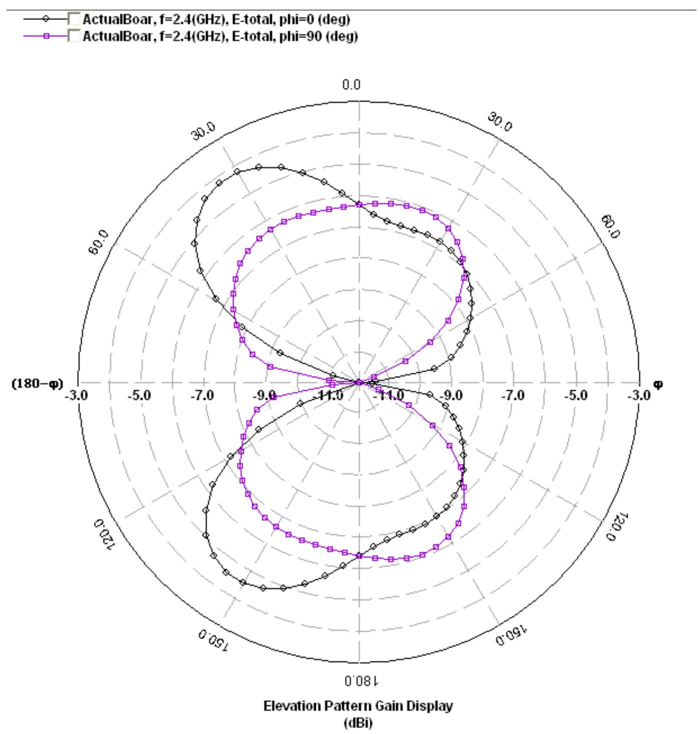


**Figure 5: E-Plane Radiation Pattern of the Actual Board**

The minimum requirement for the IFA for good performance is its return loss should be well below -10 dB at a resonant frequency of 2.44 GHz with a minimum bandwidth of 400 MHz. From the above graph it is clear that the return loss obtained from simulation at 2.44 GHz is -6.36 dB, which doesn't satisfy our requirement. Further the fabricated sample board is measured using Credence ASL 3000 RF tester for the same frequency parameters used in simulation and the results are not as expected. Measurements on the fabricated board report a return loss of -8.7 dB which is also not below -10 dB but -11dB return loss is observed at 2.6GHz range. The comparison between measured and simulated results is shown below:

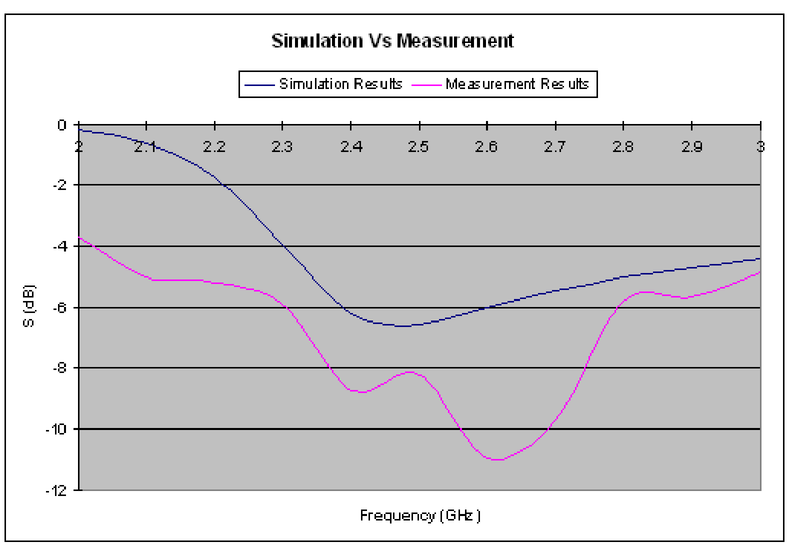


**Figure 6: The comparison between simulation and measurement**

The tester modules and the supporting PUF material affect the actual return loss measurement comparing the real environment where the WSN unit is going to be mounted. The measurement setup is shown below:

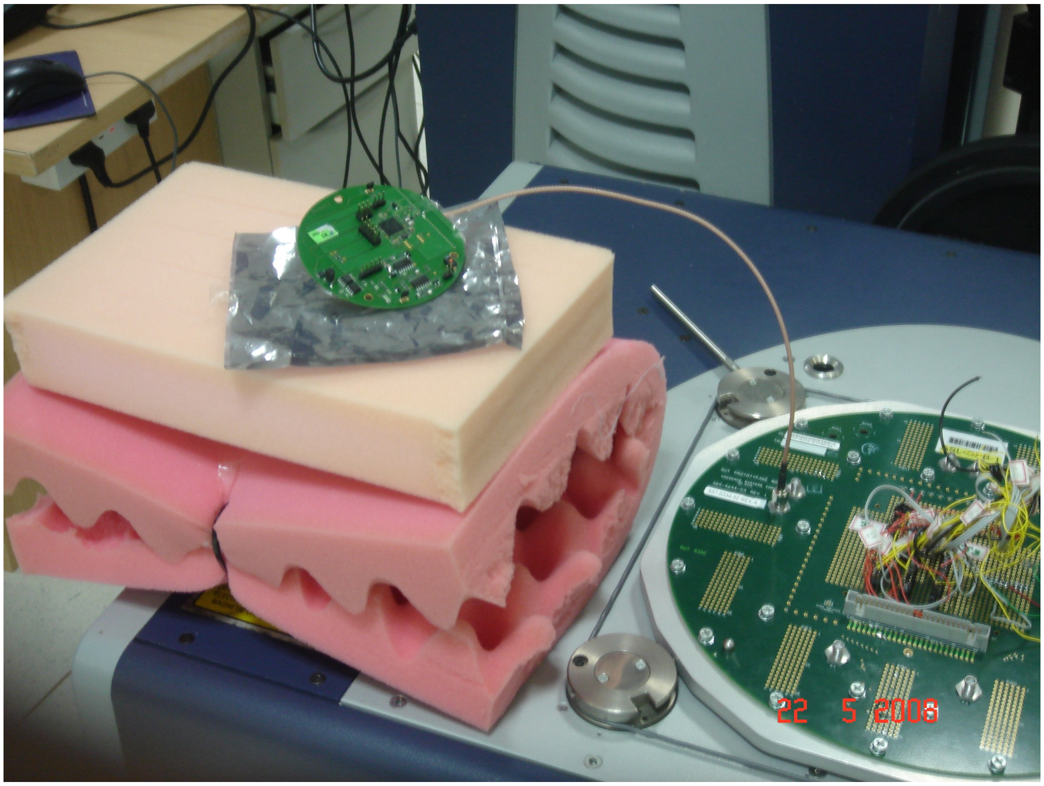


**Figure 7: The measurement setup**

**Effect of Inner plane layers on Return Loss:**

The inner two layers of the WSN PCB are Ground and Vcc respectively. They serve as return paths for the various signals and feeds power to the components present in the board. The electromagnetic field radiation and an image of itself are formed on IFA and the adjacent ground plane, respectively. So, the presence of plane in the inner layers nearer to the IFA may affect the formation of the image antenna, thus altering the field formation and affects the return loss.

In the WSN PCB we simulated, the inner plane layers are extended till the edge of the IFA as like the extension of top and bottom ground layers. We have adjusted the planes in the inner layers and simulated the return loss for different values of recessing the inner planes away from the antenna to get optimum value.

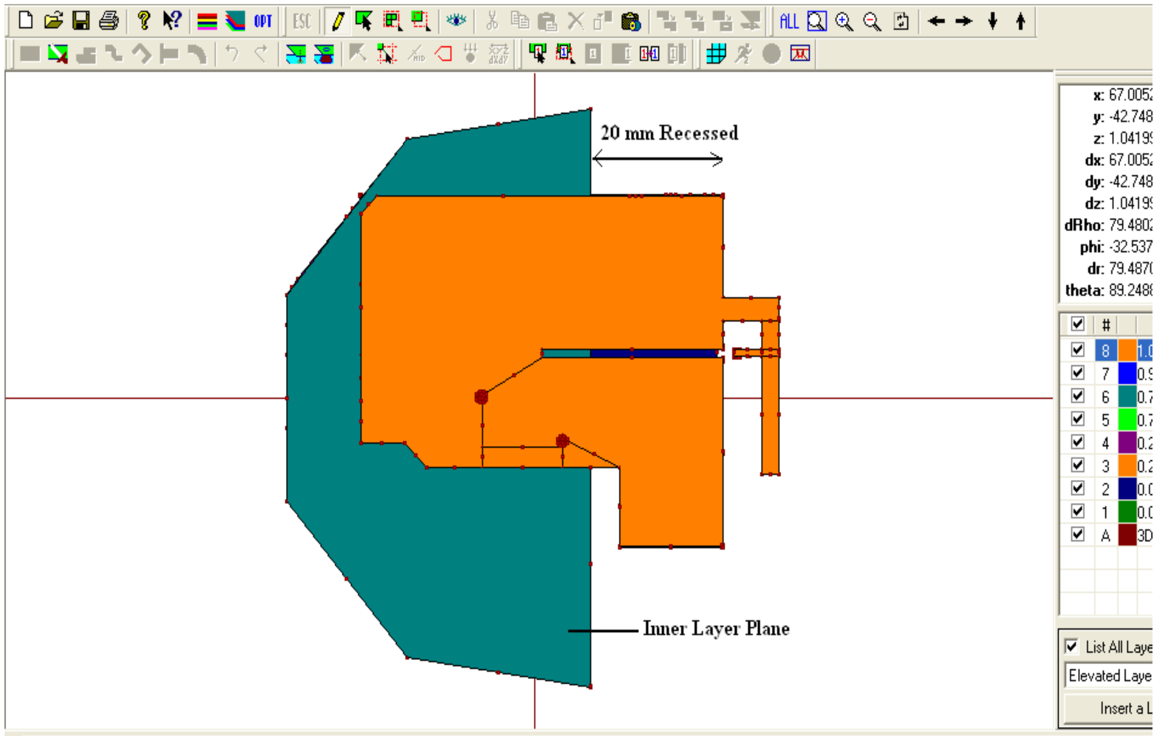


**Figure 8: The Layout file with 20mm reduced plane**

The structure shown in Figure 8 is simulated with the same frequency parameters used for the original sample board simulation. The simulation results obtained for a frequency range of 2 to 3 GHz and the comparison with the results obtained from the actual board is shown below:

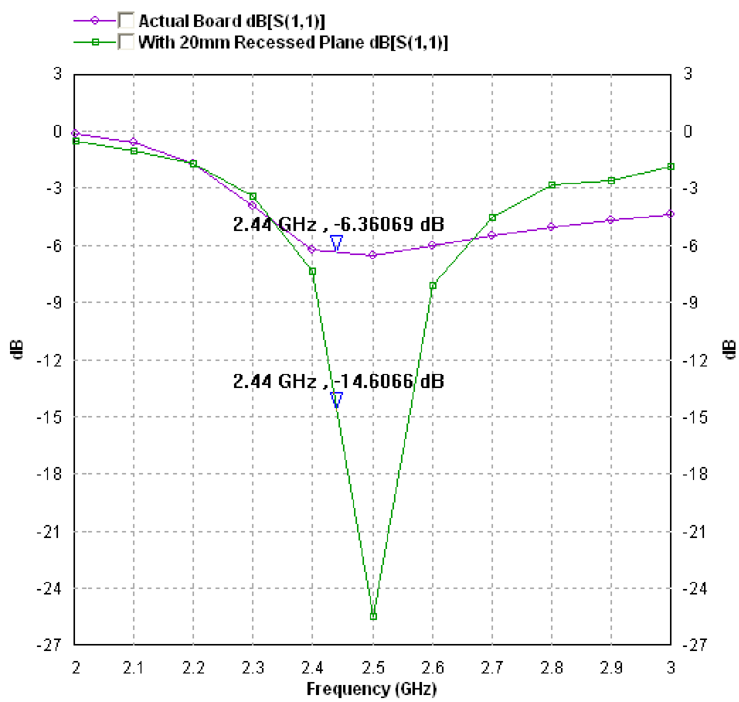


**Figure 9: Comparison of the results obtained from the actual and modified board**

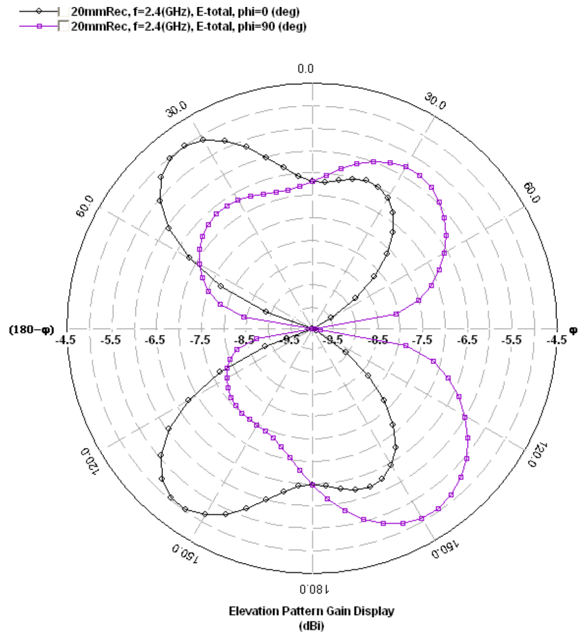


**Figure 10: E-Plane Radiation Pattern of 20mm Plane Recession Board**

The result shown in Figure 9, clearly infers that in the sample board, the dimensions and the distance to which inner plane layers both ground and Vcc are away from the IFA plays a significant part in determining the antenna's loss. The actual board simulation yielded a return loss of -6.36dB and modification in the inner layers yields a loss of -14.6066 dB at 2.44 GHz, which is far better than our basic requirement of -10 dB and also at 2.5GHz, return loss is below -24dB. Reducing the plane dimension to 20 mm yielded an appreciable result but even better result may be achieved by simulating the board with varying recessions of the planes from the antenna.

Figure 11 shows the comparison of simulation results obtained for different recessions of the ground and Vcc layers. For comparison, results obtained for recession values like 18mm, 20mm and 22mm are taken for consideration.

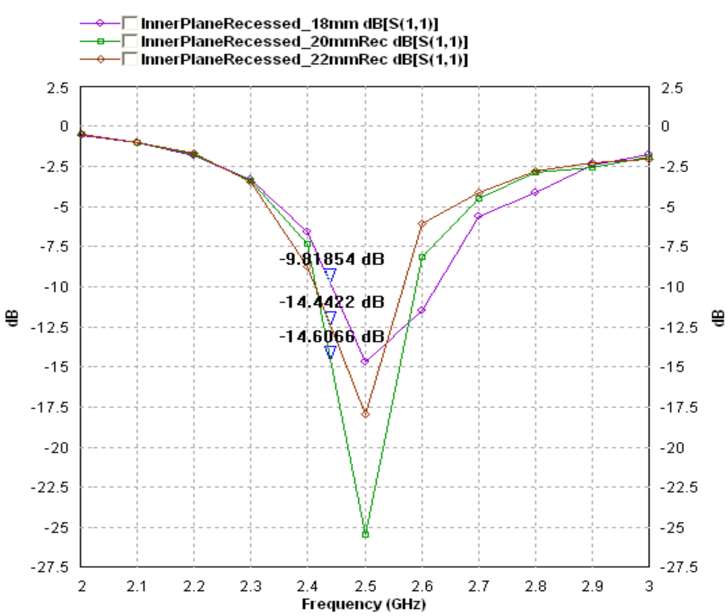


**Figure 11: Return Loss for Inner Layer recession with 18mm, 20mm and 22mm**

From the Figure 11, it is clear that increasing or decreasing the recession value above or below 20mm doesn't bring out better results than obtained for the 20mm value. The loss attained for 18mm recession is -9.81854 dB which violates our requirement and the loss obtained for 22mm recession is -14.4422 dB but not better than that obtained for 20mm. So the determination from the results and comparison is that the inner layers can be recessed to a value of 20mm to obtain desired loss values.

**Simulation of Two Layer sample Board:**

The 13213-SRB from Freescale has a PCB build of both 2 Layers and 4 Layers. The four layer SRB is named as FSL563-3. The sample board is actually designed by considering Freescale's FSL563-3 board as the reference design. The measured result of Freescale's IFA on two layer fabricated 13213-SRB found in their Application Note AN2731 is shown below:

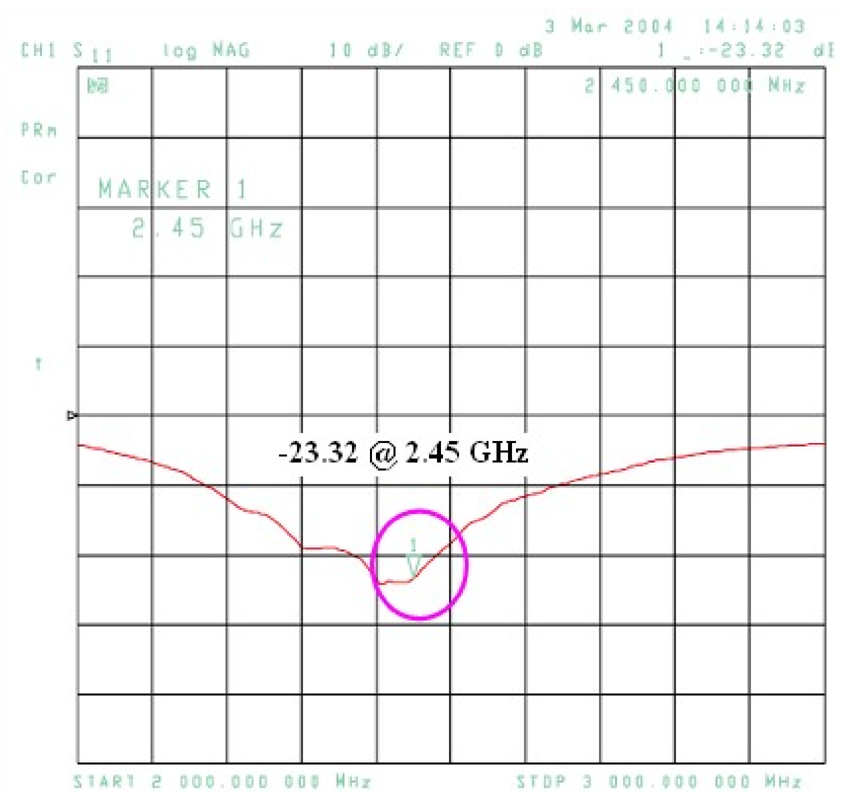


**Figure 12: Measured Return Loss found in Freescale's Application Note AN2731**

The above graph infers that the return loss of two layer Freescale board measured at 2.45 GHz is -23.32 dB. The measurement on the fabricated sample board reports a return loss of -8.7 dB and this is very high value comparing -23.32 dB.

The two layer geometry with rectangular ground shape, considering the Freescale's 13213-SRB board is drawn and simulated for return loss. The rectangular geometry for two layer board is shown below:

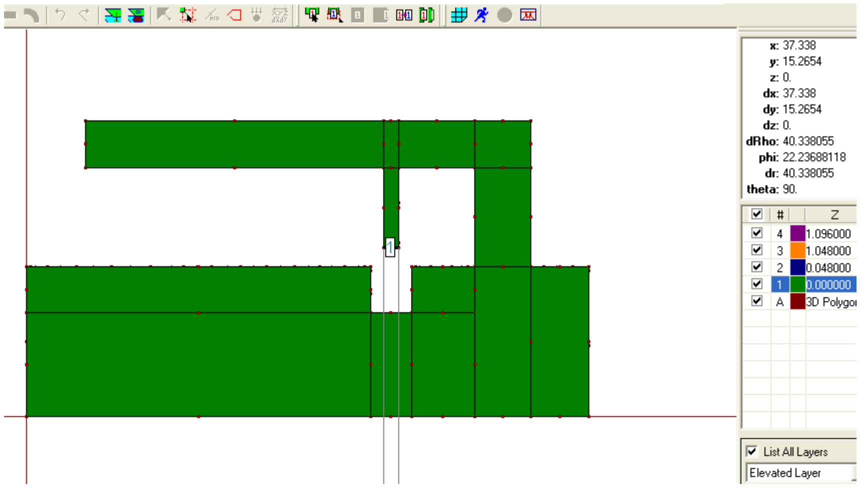


**Figure 13: Rectangular geometry for Freescale's 13213-SRB board**

The simulated result for return loss of this two layer geometry yields -16.8 dB at 2.44 GHz and is shown below:

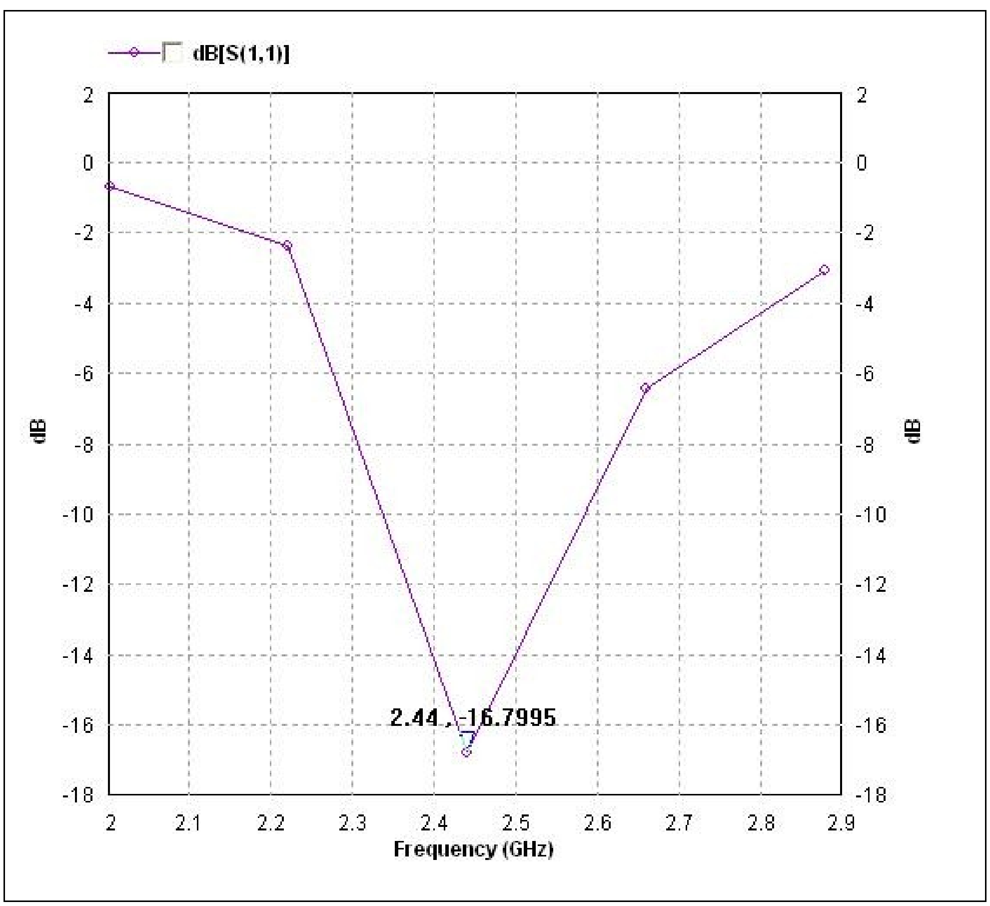


**Figure 14: The simulation output for return loss of the two layer geometry**

The four layer sample board has been modified as a two layer board and the stackup followed is shown below:

| Layer # | Layer Name | Material Description | Thickness (mm) |
|---|---|---|---|
| 1 | **Top** | Copper | 0.048 |
| | Prepreg | FR4 | 1 |
| 2 | **Bottom** | Copper | 0.048 |

**Table 2: Stack-up of the two layer PCB**

The two layer sample board is constructed in MGrid Layout Editor with the above stackup and simulated with a Meshing frequency of 20 GHz and for a frequency range of 2 to 3 GHz with 11 frequency points at a step frequency of 100 MHz. The simulation result is shown below:

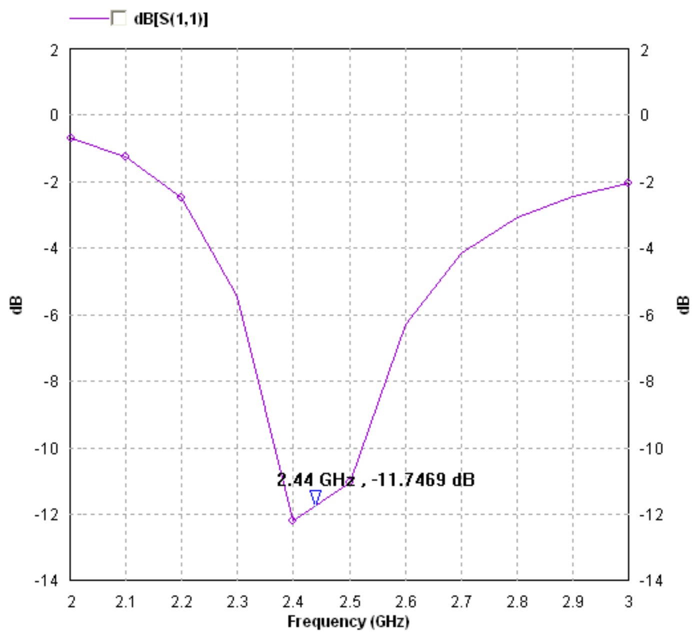


**Figure 15: Return Loss Simulation output of the two layer WSN PCB**

The two layer sample board gives a return loss of -11.7469 dB at 2.44 GHz. Comparing the two layer sample board and the four layer board with modified inner layers, the four layer board yields good return loss of -14.6066 dB at 2.44 GHz.

**Conclusion:**

Thus the performance and return loss parameters of the Inverted-F Antenna printed on both top and bottom layers of a WSN PCB has been analyzed using Zeland's IE3D simulation tool and compared with measured return loss using Credence ASL 3000 RF tester. The four layer PCB with modification in inner layers and fine tuning the antenna geometry yields reasonably very low return loss in simulation. This study show, the redesigning of the WSN PCB layout and verification by simulation will yield better performance of the printed-IFA.