Antenna Environment Impacts Efficiency and Radiation Pattern

By Gerald R. DeJean and Sean Mercer
Microsoft Corporation

In this article, the authors show the effects of location and surrounding objects on antenna performance, demonstrating the need to include analysis of antenna placement early in the design process

In this paper we have used the relatively simple planar inverted-F antenna (PIFA) [1] at 2.4 GHz to illustrate the impact of an antenna's environment on its performance. The important impacts of circuit board area, component placement and proximity of conducting surfaces are often ignored until late in a product design cycle when changes are costly and schedule impact is most painful.

We demonstrate that antenna placement and PCB geometry have a critical impact on gain and radiation patterns. With this insight, to ensure optimum outcomes, we encourage engineers to include antenna design considerations in the earliest phase of wireless product planning.

The Ground Plane is Part of the Antenna

Look at the printed circuit board shown in Figure 1. There is a PIFA antenna element (labeled "driven element") in the top right corner of the board. Most people assume, incorrectly, that the PIFA antenna element shown in the corner of this board is "the antenna." In reality the antenna is made up of both the driven element and the ground plane (shown covering the majority of the circuit board in Figure 1) [2].
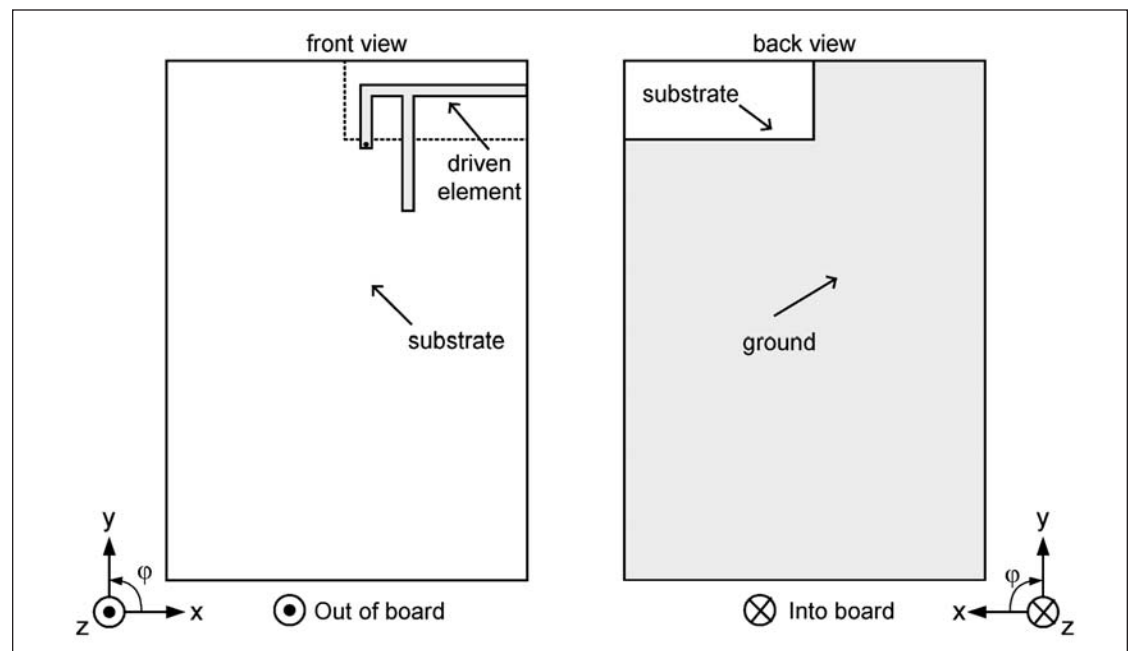
Changes in ground plane dimensions will

Figure 1 · Circuit board with antenna element.

Antenna Variable	Dimension [inches]
L_G	2
W_G	2
L_E	0.98
L_F	0.75
L_S	0.46
d_G	0.10

Table 1 · Antenna dimensions used for the electro-magnetic simulation.

Length of ground plane L_G (inches)	Peak Gain on +z-axis [dBi]
0.5	0.3
1	1.3
1.5	1.4

Table 2 · Impact of ground plane size on antenna gain.

Figure 2 · Size of ground plane influence antenna gain.

Figure 3 · PCB with PIFA simulated for initial baseline performance.

impact the gain of the antenna. Figure 2 shows the same PIFA element implemented on two different sized circuit boards. The antenna gain can be different for these two boards. Figure 2A, with the larger ground plane, will have a higher antenna gain than the smaller board shown in Figure 2B, even though the driven PIFA elements have identical dimensions in both examples.

Figure 3 shows an exploded view the PIFA element. The values of the critical dimensions shown in Figure 3 are listed in Table 1 for reference. We used an electro-magnetic simulator (CST Microwave Studio) to determine the gain of the circuit board shown in Figure 3.

The length of the circuit board ground plane L_G, shown in Figure 3, was varied while evaluating the peak antenna gain in the +z-axis. Table 2 illustrates the change in peak antenna gain as the length of the circuit board ground plane, L_G, is varied. As shown in Table 2, we determined that the circuit configuration had a maximum gain of 1.4 dBi, for a ground plane length, L_G, of 1.5 inches. Table 2 illustrates the decrease in peak gain on the +z-axis as the ground plane is reduced in size.

Radiation Pattern is Affected by Antenna Placement

Antenna related design decisions should be included as early as possible in the product development flow. As a rule of thumb, antenna radiation efficiency can be optimized by placing the antenna element close to a corner of the printed circuit board.

The driven antenna element can be placed in the corner of a PCB and the adjacent board area can be utilized for other circuit components as shown in Figure 4A. This may result in a more efficient use of PCB area than the
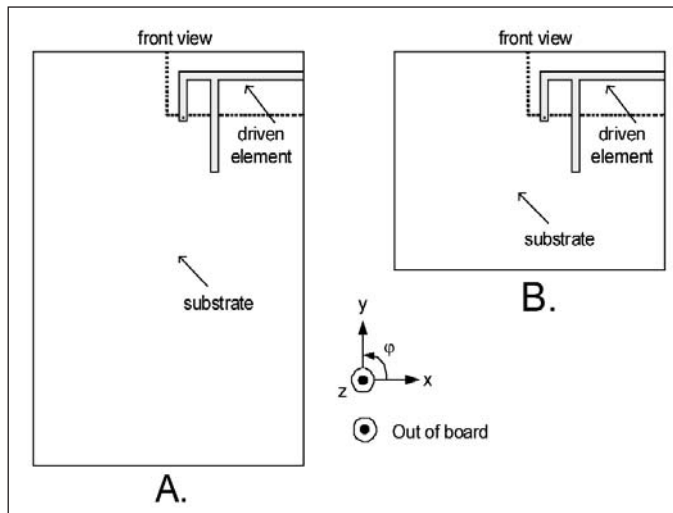
design illustrated in Figure 4B, where the entire width of the PCB above the PIFA element is reserved for the driven element and cleared of ground plane.

The antenna polar patterns for the examples illustrated in Figures 4A and 4B were simulated with a ground plane length, L_G, of 1 inch. The results shown in Figures 5 and 6 (in linear scale for enhanced resolution) demonstrate that both antenna designs exhibit similar levels of peak gain (high gain regions are shown in red with green indicating lower gain). When Figures 5C and 5D are compared with Figures 6C and 6D, it is observed that although the dimensions of the circuit boards are identical, the peak radiation emanates from the ground plane in Figure 5 (see Figure 4A for antenna) and from the driven antenna element in Figure 6 (see Figure 4B for anten-

na implementation). Hence, this clearly demonstrates that there is a price to be paid for implementing a very compact antenna design.

Typically, designers want to place the driven antenna element in the corner of the PCB to provide more space for other components. Figures 5C and 5D show that placing the radiating element in the corner of a PCB can lead to energy leakage to unintended areas of the PCB. On the other hand, in Figures 6C and 6D, the radiation is more concentrated directly above and below the element (away from the PCB). Consider how the product will be used when determining the placement of the driven antenna element.

Consider the Neighborhood— Other Objects Also Impact Antenna Performance

Consider a product using the design illustrated in Figure 4A. If a conductive external product housing was developed in parallel and designers assumed that a nonconductive aperture around the driven element would result in adequate antenna performance, this would be problematic. Most of the signal radiation, as illustrated in Figure 5, is from the circuit board and it may be shielded by the conductive product housing.

Antenna simulation early in the design phase, or measurement of early antenna and housing prototypes can prevent serious issues later in the product development cycle when product housing and user interface designs have been matured.

Dielectric bodies or conductive objects will impact antenna performance when placed near an antenna. Figure 7 shows an antenna board (same as Figure 4A with 1 inch ground length) separated from a metal plate by a distance d_P. The metal plate (red object) is behind the entire area of the circuit board (green object).

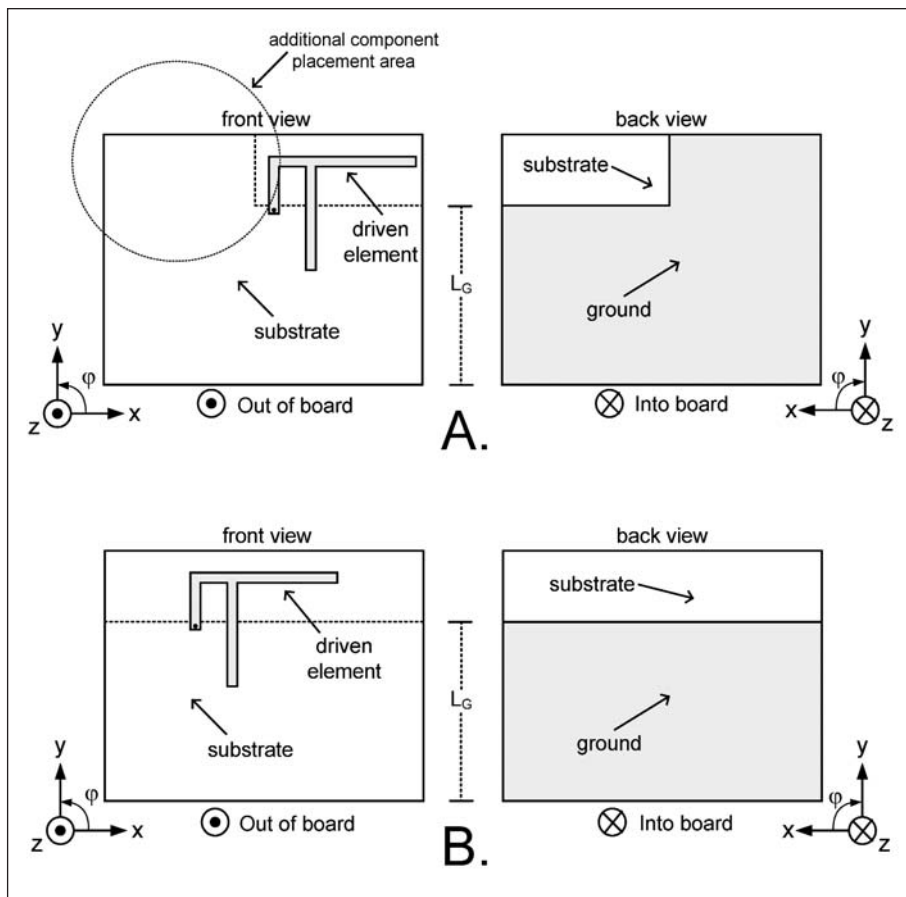The separation d_P was varied and the changes in peak antenna gain (on

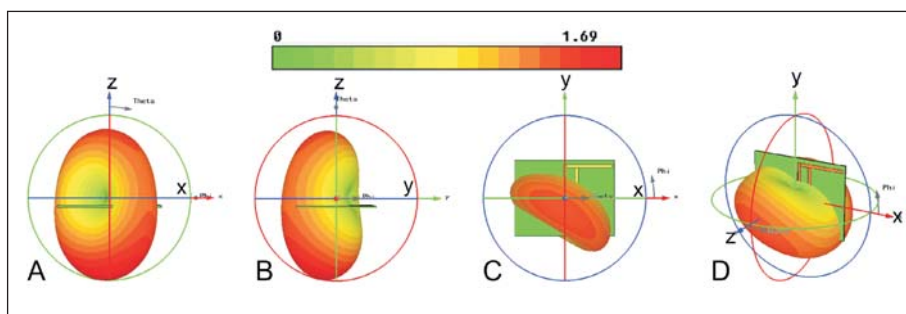**Figure 4** · **Two different ground plane geometries evaluated in the polar pattern study.**

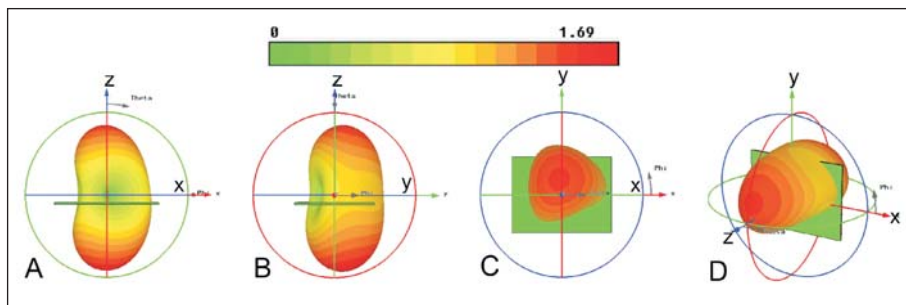**Figure 5** · **Polar patterns simulated for antenna in Figure 4a.**

Figure 6 · **Polar patterns simulated for antenna shown in Figure 4b.**

the z-axis in Figure 7) and impedance are shown in Table 3. Figure 8 illustrates the radiation patterns in linear scale when d_P = 0.01…..1.5 inches as well as the pattern for a reference antenna with no plate.

Table 3 demonstrates the impact on +z-axis peak gain variation when a metal plate is placed below the antenna. When d_P = 0.01 inch (Figure 8A), the metal plate is so close to the antenna that its radiating element acts like a microstrip line and radiation is suppressed [3].This configuration clearly has little value as an antenna

When d_P = 0.1 inch, the metal plate acts like a reflector and energy is directed upward toward space. As the metal plate separation increases from 0.1–0.5 inch, the energy becomes trapped between the metal plate and the ground plane of the antenna. As a result, the peak gain on +z-axis reaches a "radiation" minimum. (Remember that when d_P = 0.01 inch, the antenna does not radiate.)

Finally when d_P = 1 and 1.5 inches, the antenna is far enough away from the metal plate that the energy trapped between the metal plate and the antenna starts to diminish. As a result, the peak gain on +z-axis is greatly enhanced. An interesting point to note is the fact that the peak gain on +z-axis (in some instances) is greater when the metal plate is included in the analysis, even when d_P = 1.5 inch, the farthest separation in the study. This leads to a conclusion that sometimes the metal plate is also part of the antenna even though the plate is not physically connected to the antenna. Therefore, careful attention needs to be placed on analyzing the antenna's performance when objects are close to the antenna.

Figure 9B shows a circuit board (green object) placed against the surface of a metal box (red object). As expected from Table 3 (see gain with d_P = 0.01), antenna gain and match
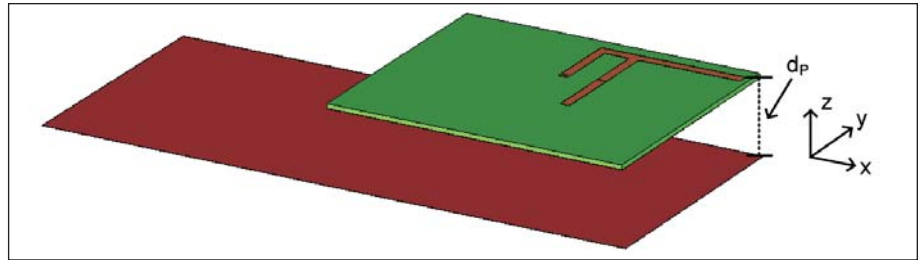

Figure 7 · Metal plate behind antenna PCB.

Figure 8 Letter	Separation of metal plate from PCB, d_P [inches]	Peak Gain on +z-axis [dBi]	Antenna Impedance [Ω]	Return Loss [dB]
A	0.01	−8.1	62.9-j*296.5	0.6
B	0.1	3.8	72.6-j*51.7	7.2
C	0.2	1.4	83.1+j*3.2	11.4
D	0.3	−0.6	92.0+j*1.2	10
E	0.4	−1.4	93.1-j*7.5	9.7
F	0.5	−1.1	90.8-j*15.1	9.7
G	1	3.3	65.7-j*33.4	9.9
H	1.5	4.6	54.1-j*26.7	11.7
I	infinity	2.3	57.8-j*17.9	14.4

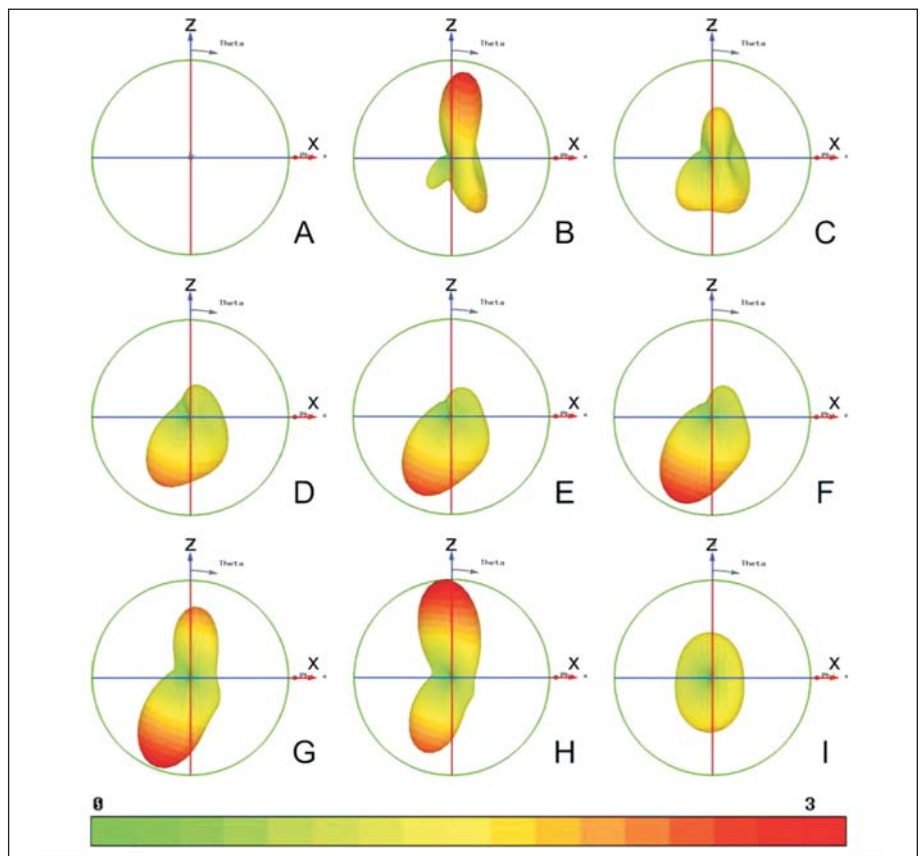Table 3 · Impact of metal plate on antenna gain.

Figure 8 · Radiation patterns when separation between metal plate and antenna (d_P) is varied from 0.01 inches to infinity

will be very poor under this condition.

One solution could be to position a portion of the circuit board beyond the edge of the metal box as illustrated in Figure 9A. In this case, the adverse impact of placing the grounded structure close to the driven antenna element is greatly diminished and the majority of the circuit board could be placed close to the metal structure if required by product geometry constraints. The data in Table 4 clearly demonstrates the performance advantage of Figure 9A over that of Figure 9B (d_B = 0.5 inch was used to be consistent with the antenna design shown in Figure 4B). The height of the metal box, h_{MB}, is 2 inches.

If product design constraints preclude the possibility of mounting the PIFA element above the top of the metal case (perhaps this would make the total height unacceptably large) per Figure 9A then another option may be to cut away a portion of the metal housing immediately behind the PIFA element. As shown in Figure 10, this can increase the separation between the driven element and the grounded metal structure.

Table 5 shows the effect of increasing the separation d_{CO} between the PIFA element and the metal box. Although, antenna gain is significantly degraded when the metal structure is close to the driven element (small values of d_{CO}), cutting away a portion of the box under the radiating element does provide some improvement in comparison to the configuration in Figure 9B in terms of peak gain on +z-axis. In addition, the antenna impedance and gain appear to stabilize once the metal plate separation reaches about 1 inch. Therefore, values of d_{CO} > 1 inch do not provide significant improvement.

Suggestions for Future Product Development

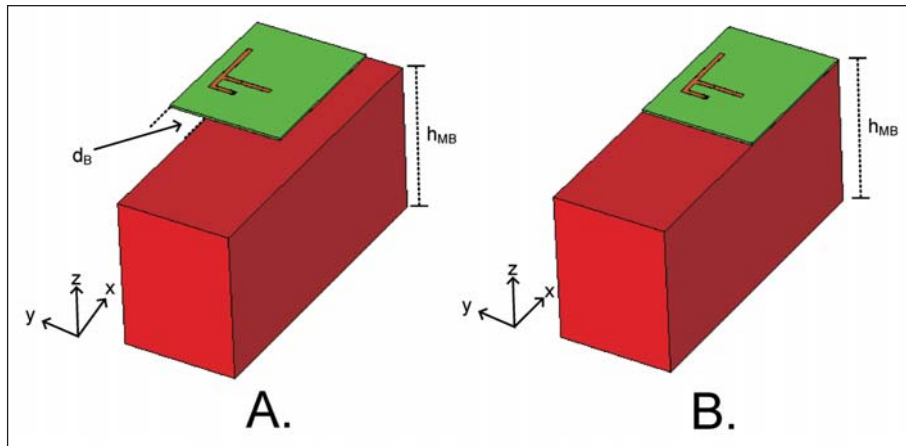We have used a 2.4 GHz PIFA antenna example to illustrate several important issues that are encountered when adding wireless functionality to a product. Although we limited our discussion (for illustration purposes) to a PIFA element, the same issues will apply to the integration of any antenna into a product. Also, similar design integration issues will be encountered at any other radio frequency.

It is important to understand that the size of a ground plane influences the performance of the antenna—the ground plane is actually part of the antenna for most antenna implementations.

Figure 9 · Diagram showing PIFA element above box.

Figure 9 Letter	Position of PIFA element above metal box d_B [inches]	Peak Gain on +z-axis [dB*i*]	Antenna Impedance [Ω]	Return Loss [dB]
B	0	−12.6	1252+j*700.5	0.5
A	0.5	0.6	31.4+j*10.5	12.24

Table 4 · Impact of moving PIFA element beyond the edge of the metal housing.
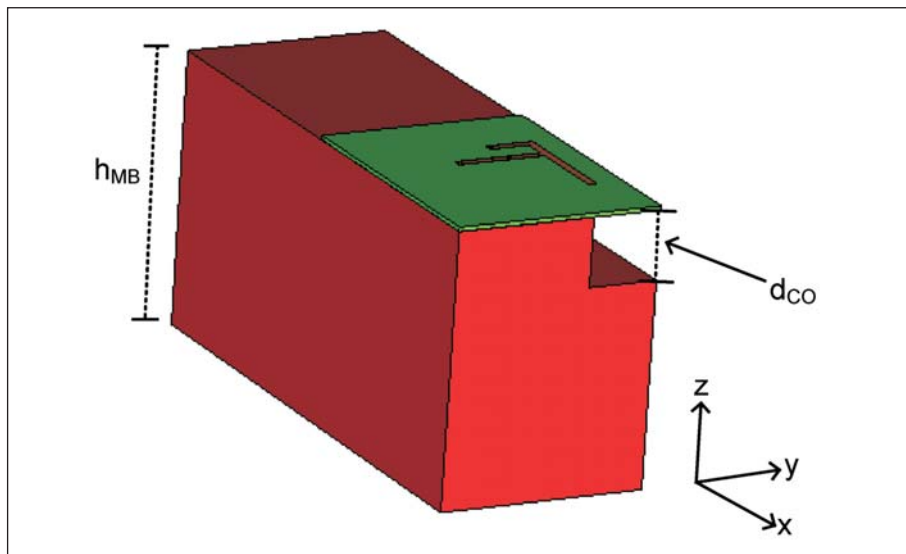

Figure 10 · Diagram showing PIFA with box cut away behind the PIFA.

Although the driven antenna element can be placed in a corner of a PCB to minimize the amount of board area dedicated to the antenna, this can impact the antenna gain and polar pattern. We have shown that placing a ground plane (or other components) adjacent to the driven element will result in an antenna design with a null in the polar pattern. Put another away, this antenna will have less gain in some product orientations than a PCB design that allocates more board area to the driven element. This may be perfectly acceptable but early design analysis may prevent unnecessary product development iterations.

We recommend considering the product's usage scenarios at the beginning of the design phase to ensure that antenna board area is assigned based on an appropriate trade-off between required perfor-

Separation of metal plate from PCB, dCO [in]	Peak Gain on +z-axis [dBi]	Antenna Impedance [Ω]	Return Loss [dB]
0.1	−6.6	$6.1+j24.4$	1.8
0.2	−5.3	$10.3+j20.4$	3.2
0.3	−4.5	$14.6+j20.8$	4.5
0.4	−3.6	$17.7+j21.3$	5.4
0.5	−3.1	$20.9+j21.3$	6.4
1	−1.5	$27.9+j17.3$	9.4
1.5	−1.5	$27.7+j14.3$	9.9

Table 5 · Impact of cutting away metal box behind antenna element.

mance and available space. Changing board dimensions or layouts later in the design phase can significantly degrade performance, prolong schedules and increase cost. Antenna simulation and/or measurement as early as possible in the product development cycle will avoid unpleasant and potentially costly surprises later on.

References

1. M. Ali & G. J. Hayes, "Analysis of integrated inverted-F antennas for Bluetooth applications," *IEEE Conference on Antennas and Propagation for Wireless Communications,* November 2000, pp. 21-24.

2. M.-C. Huynh & W. Stutzman, "Ground plane effects on planar inverted-F antenna (PIFA) perfor-

mance," *IEEE Proceedings on Microwaves, Antennas and Propagation,* Volume 150, Issue 4, August 2003, pp. 209-213.

3. R. S. Elliott, *Antenna Theory and Design,* New Jersey, John Wiley & Sons, Inc., 2003.

Author information

Gerald R. DeJean is a researcher at Microsoft Research in the field of RF and antenna design. He conducts research in antenna design, RF/microwave design and characterization, and 3D system-on-package (SOP) integration of embedded functions that focuses largely on modern commercial RF systems. He is also an adjunct assistant professor at the Georgia Institute of Technology. Gerald received the B.S. degree in Electrical and Computer Engineering from Michigan State University and the M.S. and Ph.D. degrees in Electrical and Computer Engineering from the Georgia Institute of Technology. He has authored and co-authored over 40 papers in refereed journals and conference proceedings.

Sean Mercer is a Wireless Architect at Microsoft Corporation. He received the M.Sc. (Eng.) in 1987 and a Ph.D. in 1990 from the University of Cape Town, South Africa. His past experience includes HF transceiver design, CDMA and GSM phone implementations, antenna designs and amplifier and oscillator implementations up to X-band. Sean has several past publications and has two U.S. patents issued. He maintains his CEng status in the UK and has been a MIET and MIEEE for over a decade. He can be reached at seanme@microsoft.com

---

Writing for *High Frequency Electronics*

There are a vast number of applications using the design techniques that are unique to high frequency/high speed electronic circuits and systems. We are always looking for technical articles to publish in *High Frequency Electronics,* on nearly any topic. To serve engineers at all experience levels, we include material ranging from basic tutorials to advanced techniques, with a strong emphasis on practical applications. Articles may cover circuit and system design, test methods, EDA simulation, regulatory compliance, manufacturing, or any other area that is useful to fellow engineers.

Send proposals to the Editorial Director: gary@highfrequencyelectronics.com