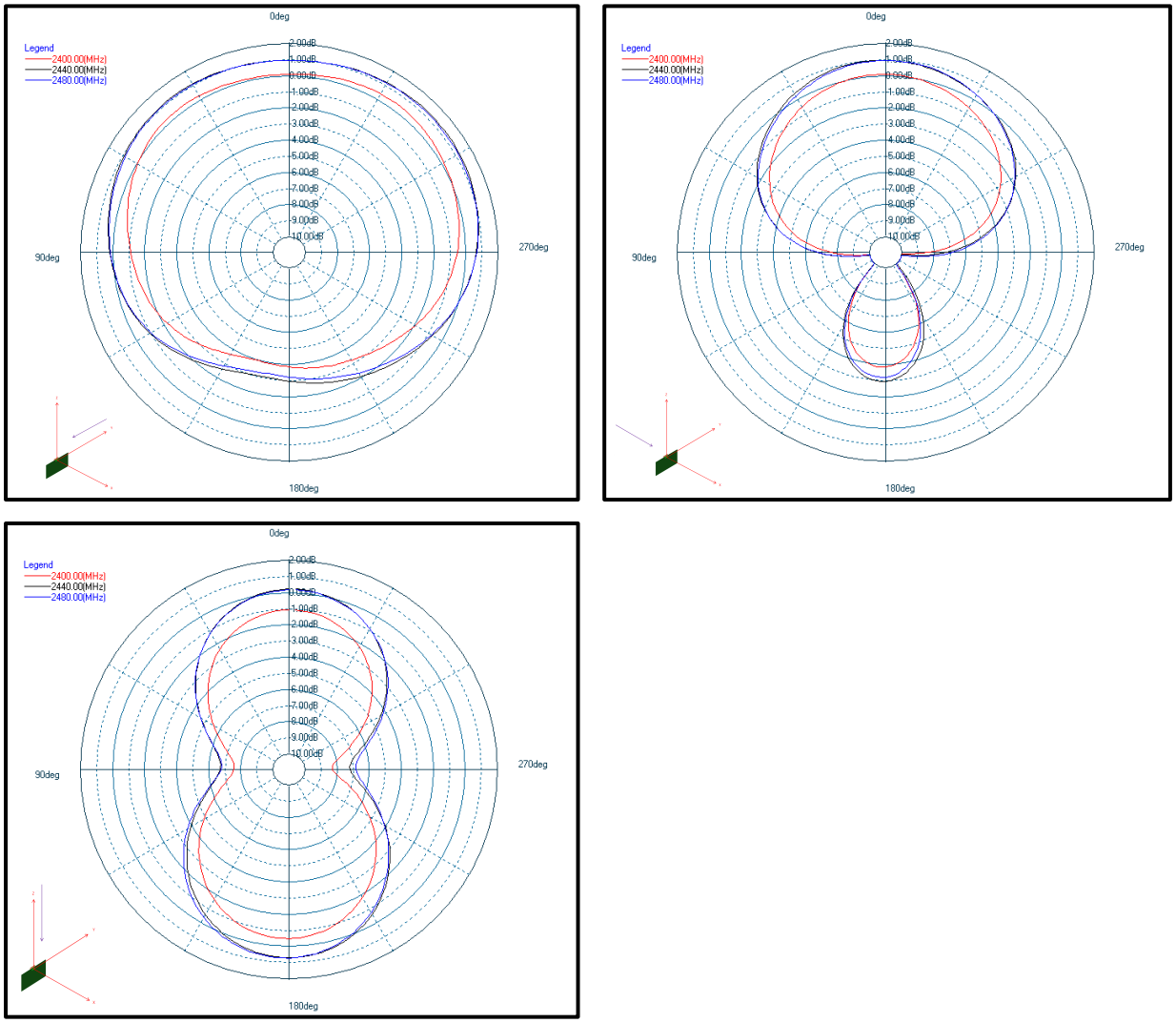
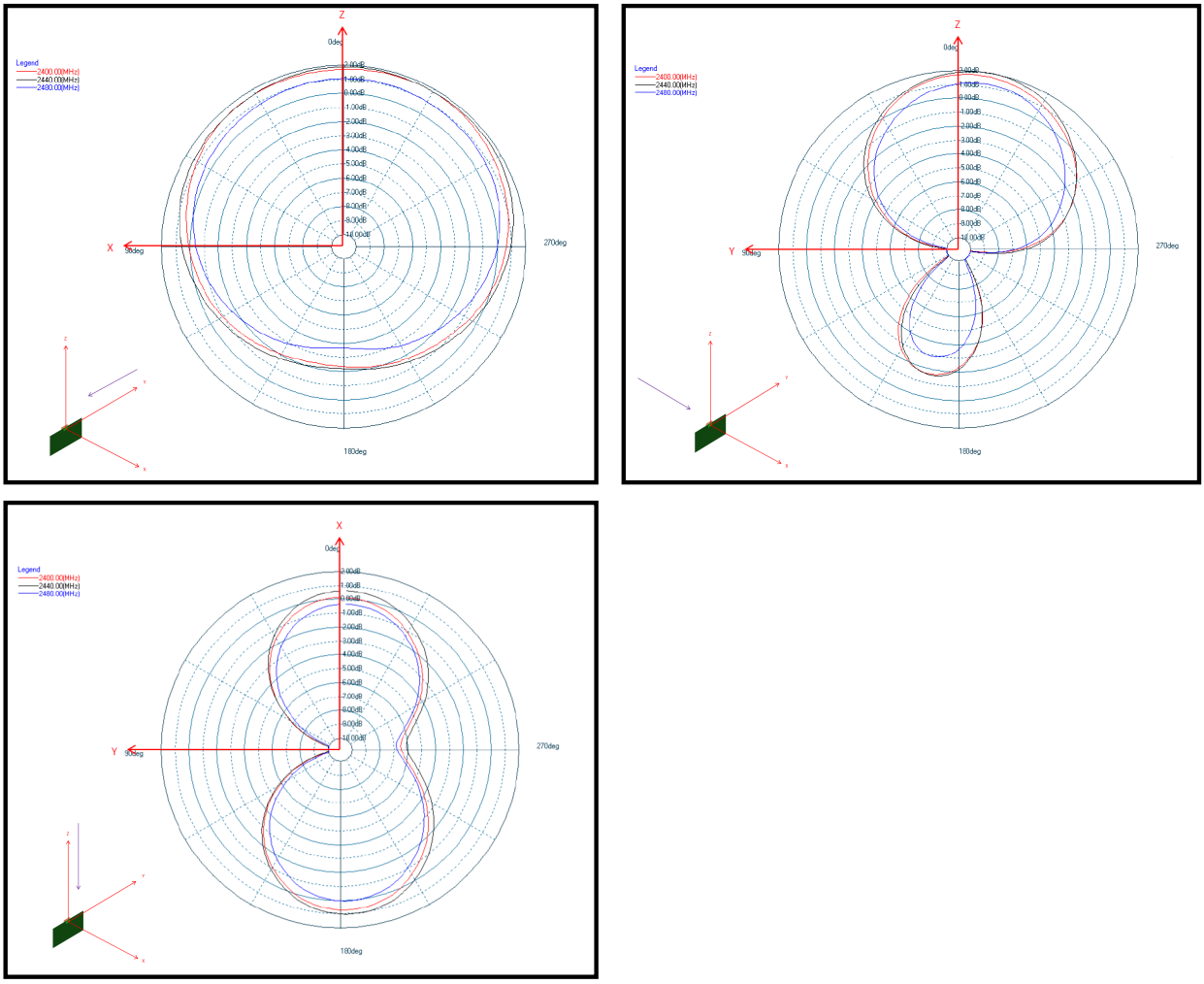
4.16.1 Antenna Typical Characteristics

Typical BGM220S radiation patterns under optimal operating conditions are plotted in the figures that follow. Antenna gain and radiation patterns have a strong dependence on the size and shape of the application PCB the module is mounted on, as well as on the proximity of any mechanical design to the antenna.



Top Left: Phi 0°, Top Right: Phi 90°, Bottom Left: Theta 90°

Figure 4.3. BGM220S12A Typical 2D Antenna Radiation Patterns on 50 mm x 30 mm board



Top Left: Phi 0º, Top Right: Phi 90º, Bottom Left: Theta 90º

Figure 4.4. BGM220S22A Typical 2D Antenna Radiation Patterns on 55 mm x 20 mm board

7. Design Guidelines

7.1 Layout and Placement

For optimal performance of the BGM220S the following guidelines are recommended:

- Place the module 1.50 mm from the edge of the copper “keep-in” area at the middle of the long edge of the application PCB, as illustrated in Figure 7.1 Recommended Layout for BGM220S on page 32.
- Copy the exact antenna design from Figure 7.2 Antenna Layout With Coordinates on page 33 with the values for coordinates A to L given in Table 7.1 Antenna Polygon Coordinates, Referenced to Center of BGM220S on page 33.
- Make a cutout in all lower layers aligned with the right edge and the bottom edge of the antenna as indicated by the yellow box in Figure 7.3 Antenna Clearance in Inner and Bottom Layers on page 34.
- Connect all ground pads directly to a solid ground plane in the top layer.
- Connect RF_2G4 to ANT_IN through a 0-ohm resistor.
  - The 0-ohm gives the ability to test conducted and to evaluate the antenna impedance in the design.
- Place ground vias as close to the ground pads of the BGM220S as possible.
- Place ground vias along the antenna loop right and bottom side.
- Place ground vias along the edges of the application board.
- Do not place plastic or any other dielectric material in contact with the antenna.
  - A minimum clearance of 0.5 mm is advised.
  - Solder mask, conformal coating and other thin dielectric layers are acceptable directly on top of the antenna region.

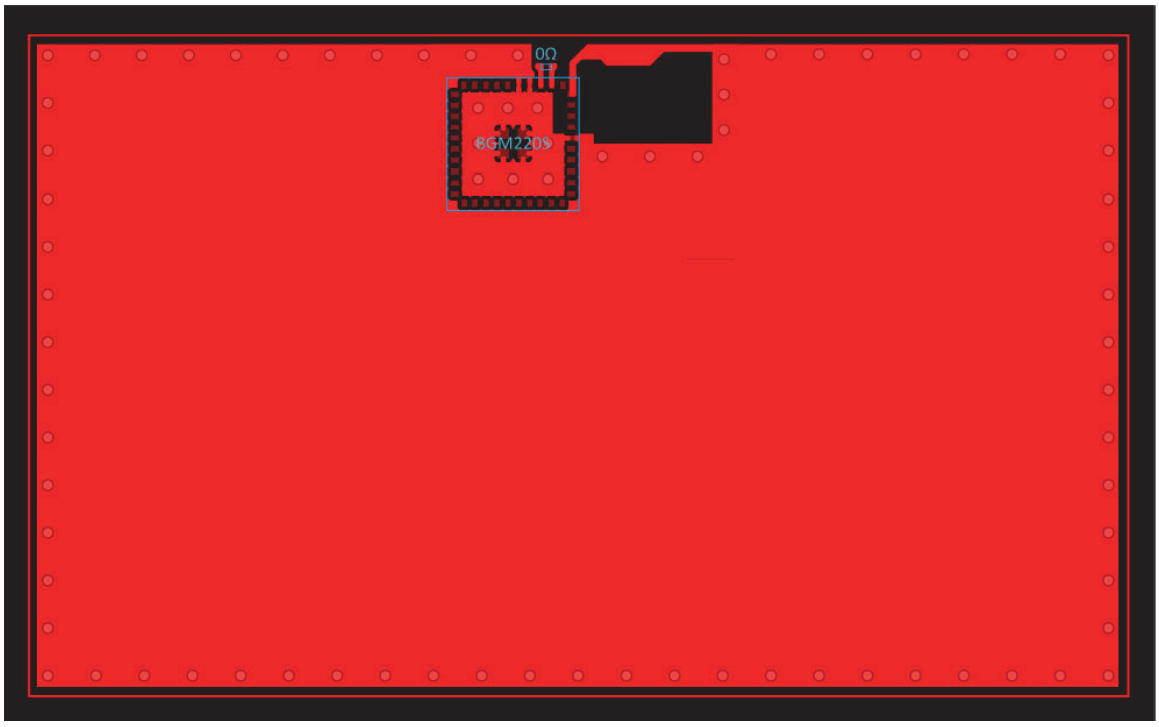


Figure 7.1. Recommended Layout for BGM220S

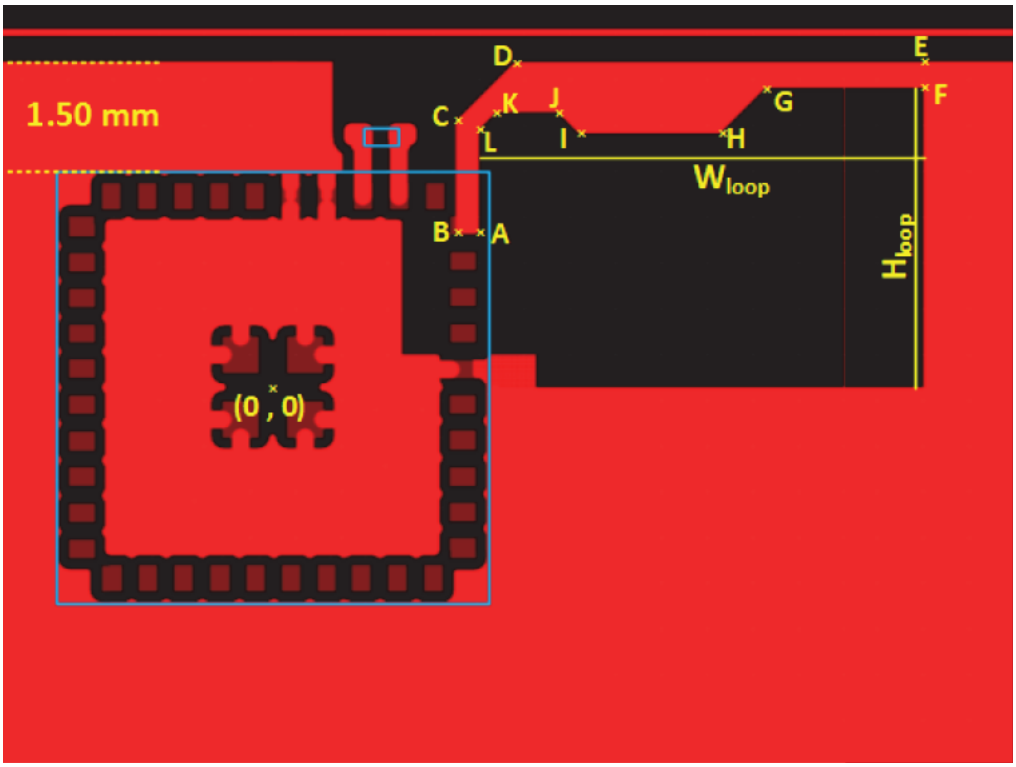


Figure 7.2. Antenna Layout With Coordinates

Table 7.1. Antenna Polygon Coordinates, Referenced to Center of BGM220S

Point	BGM220SC12WGA2	BGM220SC22WGA2 BGM220SC22HNA2
A	(2.87, 2.13)	(2.87, 2.13)
B	(2.54, 2.13)	(2.54, 2.13)
C	(2.54, 3.69)	(2.54, 3.69)
D	(3.36, 4.51)	(3.36, 4.51)
E	(8.85, 4.51)	(7.75, 4.51)
F	(8.85, 4.15)	(7.75, 4.15)
G	(6.84, 4.15)	(6.84, 4.15)
H	(6.21, 3.52)	(6.21, 3.52)
I	(4.26, 3.52)	(4.26, 3.52)
J	(3.97, 3.81)	(3.97, 3.81)
K	(3.10, 3.81)	(3.10, 3.81)
L	(2.87, 3.58)	(2.87, 3.58)
W_{loop}	5.98	4.88
H_{loop}	4.15	4.15

Note:
1. All coordinates and dimensions listed in mm.

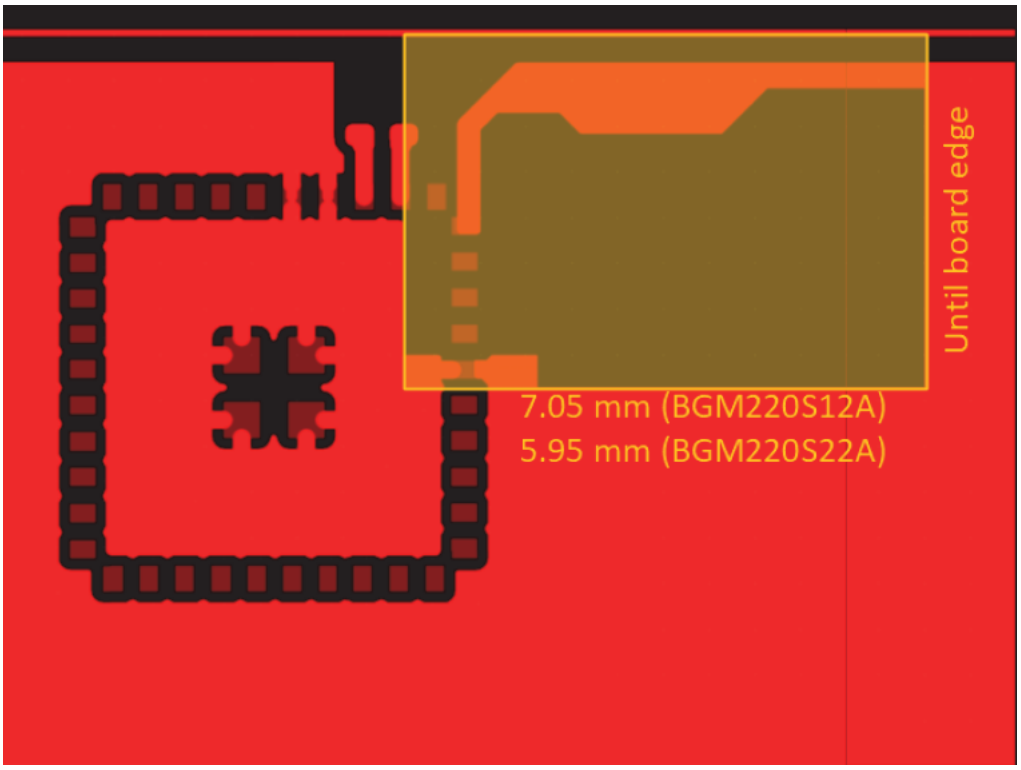


Figure 7.3. Antenna Clearance in Inner and Bottom Layers

7.2 Best Design Practices

The design of a good RF system relies on thoughtful placement and routing of the RF signals. The following guidelines are recommended:

- Place the BGM220S and antenna close to the center of the longest edge of the application board.
- Do not place any circuitry between the board edge and the antenna.
- Make sure to tie all GND planes in the application board together with as many vias as can be fitted.
- Generally ground planes are recommended in all areas of the application board except in the antenna keep-out area shown in Figure 7.3 Antenna Clearance in Inner and Bottom Layers on page 34.
- Open-ended stubs of copper in the outer layer ground planes must be removed if they are more than 5 mm long to avoid radiation of spurious emissions.
- The width of the GND plane to the sides of the BGM220S will impact the efficiency of the on-board chip antenna.
  - To achieve optimal performance, a GND plane width of 50 mm for BGM220S12A or 55 mm for BGM220S22A is recommended as seen on Figure 7.4 Illustration of Recommended Board Width on page 35.
  - See 4.16.1 Antenna Typical Characteristics for reference.

Figure 7.5 Non-Optimal Layout Examples on page 36 illustrates layout scenarios that will lead to severely degraded RF performance for the application board.

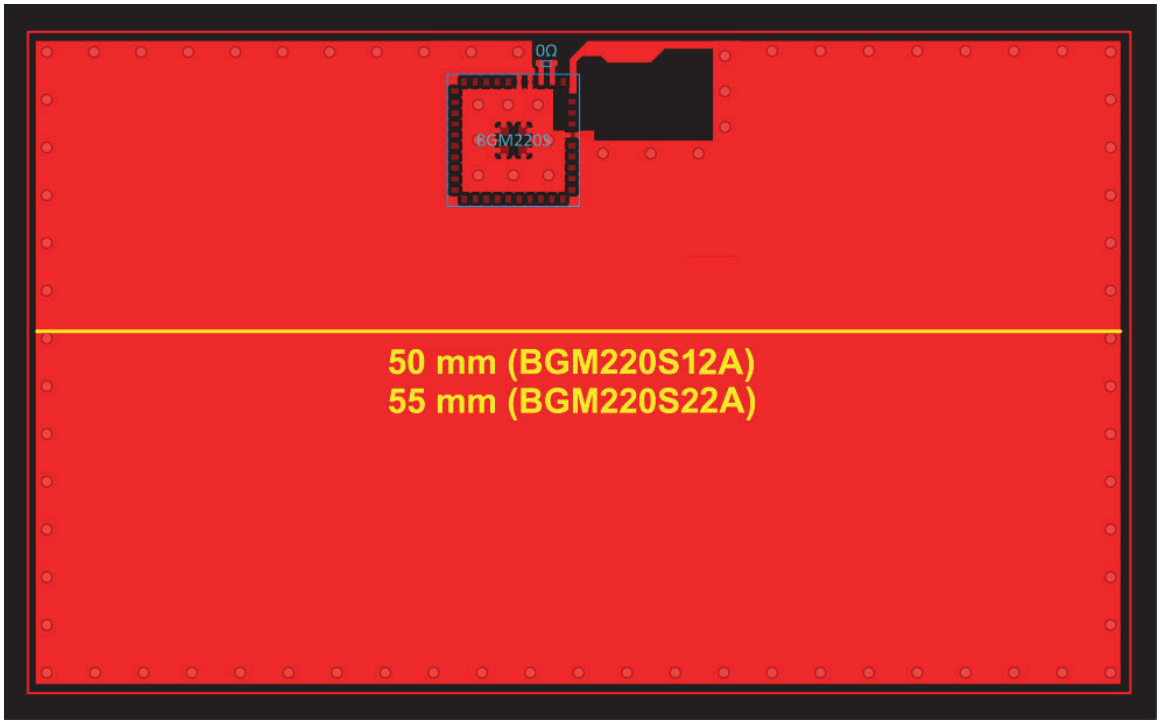


Figure 7.4. Illustration of Recommended Board Width

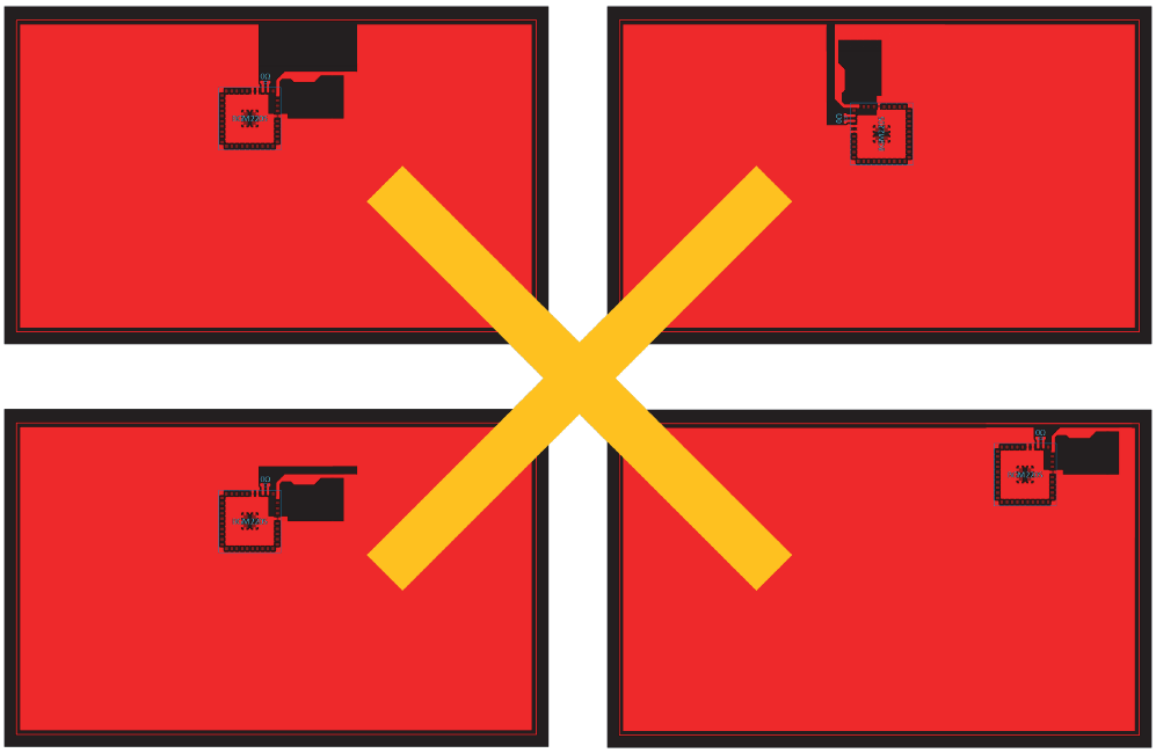


Figure 7.5. Non-Optimal Layout Examples

7.3 Radio Performance vs. Carrier Board Size

For many applications, the carrier board size is determined by the overall form factor or size of the additional circuitry. The recommended carrier board width of 50 mm for the BGM220S12A and 55 mm for the BGM220S22A is thus not always possible in the end-application. If another form factor is required, the antenna performance of the integrated antenna will be compromised but it may still be sufficiently good for providing the required link quality and range of the end-application. Figure 7.6 Efficiency of the Integrated Antenna as Function of the Carrier Board Size for BGM220S12A on page 37 and Figure 7.7 Efficiency of the Integrated Antenna as Function of the Carrier Board Size for BGM220S22A on page 38 show the total efficiency of the integrated antenna for different carrier board sizes. As can be seen the best performance is achieved for the carrier board size of 50 mm x 30 mm for the BGM220S12A and 55 mm x 25 mm for the BGM220S22A, with relatively constant performance for larger boards and rapidly declining performance for smaller boards.

The performance of all the sizes tested will be adequate for more than 15 m line-of-sight range and all of the sizes are thus usable.

WARNING: Any antenna tuning or change of the loop dimensions will void the modular certification of modules with modular certification. In that case, a Permissive Change to the modular approval is required.

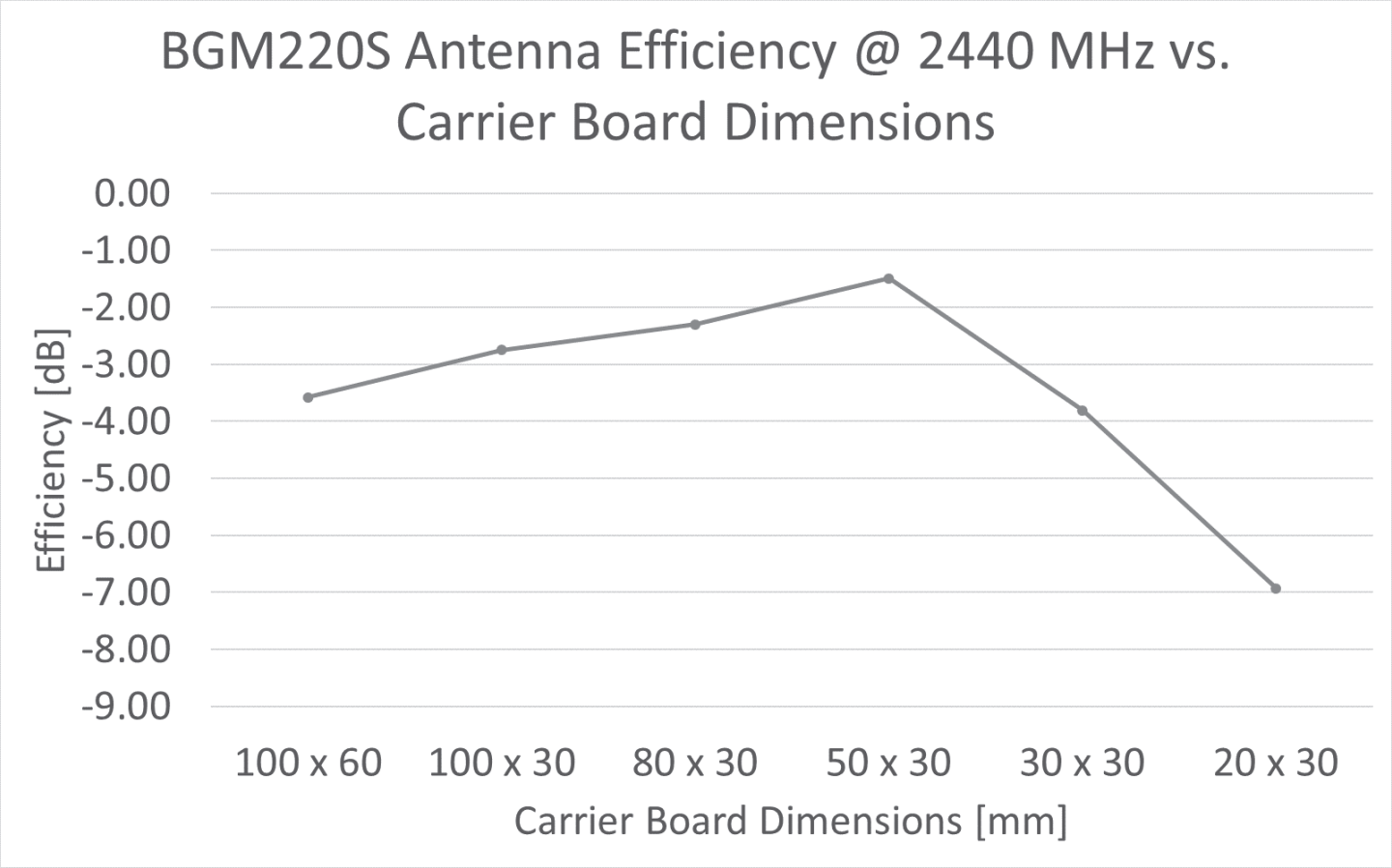


Figure 7.6. Efficiency of the Integrated Antenna as Function of the Carrier Board Size for BGM220S12A

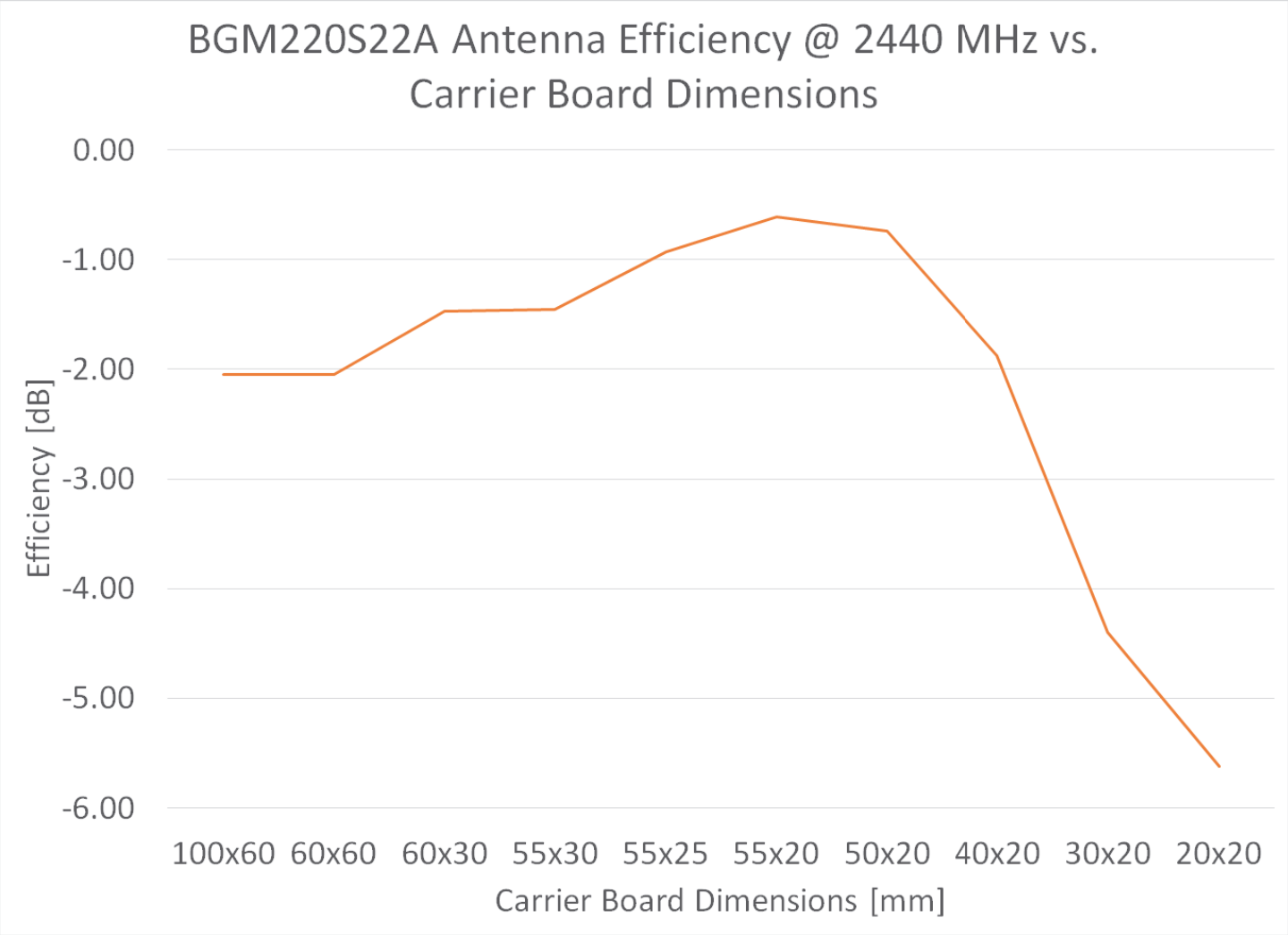


Figure 7.7. Efficiency of the Integrated Antenna as Function of the Carrier Board Size for BGM220S22A

7.4 Proximity to Other Materials

Placing plastic or any other dielectric material directly in contact with the antenna may cause performance degradation. A clearance of minimum 0.5 mm is recommended to avoid excessive detuning of the antenna. Solder mask, conformal coating, and other thin dielectric layers are acceptable directly on top of the antenna region. Any metallic objects in close proximity to the antenna will prevent the antenna from radiating freely. The minimum recommended distance of metallic and/or conductive objects is 10 mm in any direction from the antenna except in the directions of the application PCB ground planes.

7.5 Proximity to Human Body

Placing the module in contact with or very close to the human body will negatively impact antenna efficiency and reduce range. Furthermore, additional certification may be required if the module is used in a wearable device.