AN1394: BGM220S Antenna Tuning Guide

This application note provides guidelines on how to design PCBs for BGM220S modules and how to tune the module integrated antenna.

The information provided is supplemental to the design guidelines and recommendations included in the BGM220S data sheet.

The main purpose of this document is to provide additional design considerations and best practices for use-cases when exact implementation of the reference design is not feasible.

KEY FEATURES

- Layout guidelines
- Antenna tuning guidelines
- Antenna efficiency reference values

1. Device Compatibility

This application note supports the following devices:

- EFR32BG22-based SiP Modules:
  - BGM220SC12
  - BGM220SC22

2. Introduction

Antennas (by their nature) are sensitive to the electromagnetic characteristics of their environment and therefore an antenna which was optimized for a given use-case may become detuned in a different application. While the BGM220S integrated antenna is relatively robust to detuning effects, its performance still can be subject to some degree of degradation if the application significantly deviates from the reference design and these differences are not compensated appropriately.

Furthermore, the BGM220S antenna is also prone to part-to-part variation due to component spreading (see 8. Appendix 3 – Part-to-Part Variation for more details) and therefore it is recommended to make sure there is adequate margin in the antenna matching to account for this part-to-part spreading.

In a typical application, it is usually the host PCB shape and stack-up related recommendations which are most difficult to implement because these parameters are often bounded by other application requirements as well. Therefore, the primary objective of this document is to provide additional information and recommendations on how to maximize the antenna performance even when the carrier board specifications are differing from the optimum reference design.

3. Generic RF Layout Considerations

- Place the BGM220S at the center of a PCB edge with ~55 mm length. The center line of the first antenna trace (which is perpendicular to the board edge) should coincide with the PCB edge center line (as shown in Figure 4.1 Optimum PCB Dimensions and Module Placement on page 5).
- Avoid placing any components or routing in the close proximity of the antenna loop area.
- Unless otherwise specified, ground pours are recommended in all areas of the application board. Restrict GND metal keep-outs to the internal areas of the PCB and keep the board GND metallization outline intact.
- Avoid separation of the ground plane metallization (especially between the BMG220S and the antenna loop) and use as many GND stitching vias as possible between the GND planes on different layers.
- Add GND stitching vias near the GND pins and along the antenna loop to minimize series parasitic inductance between the GND pins and the ground planes of different layers.
- Use as many GND stitching vias as possible along the PCB edges. The maximum distance between the vias at the board edges should be less than lambda/10 of the 10th harmonic (the typical distance between vias on reference radio boards is 40–50 mil). This distance is required to reduce the PCB radiation at higher harmonics caused by the fringing field.
- For designs with more than two layers, it is recommended to put as many traces (even the digital traces) as possible in an inner layer and ensure large, continuous GND pours on the top and bottom layers.
- Avoid using loops and long wires, especially near the PCB edges as these could also become a source of unwanted spurious radiation.
- Use a 4-layer PCB stack-up with 1.6 mm total thickness and ~4.4 relative permittivity (a.k.a. dielectric constant). The stack-up used in the reference designs is detailed in the figure below.

```
BOARD THICKNESS : 1.6 mm +/-10%
NO OF LAYERS    : 4
MATERIAL(S)     : Glass Epoxy FR-4, NEMA Class 2, UL 94V-0, Tg min 150 C
                  Materials in compliance with the RoHS and WEEE directives
BUILD UP        :
                TOP ====| |========================= 38 um Cu (ca) After plating
                   /////| |// PREPREG or CORE ////// 300um *
                L1 =====| |========================= 18 um Cu (0.5 Oz)
                   /////| |// PREPREG or CORE ////// **
                L2 =====| |========================= 18 um Cu (0.5 Oz)
                   - - -| |// PREPREG or CORE ////// 300um *
                BOT ====| |========================= 38 um Cu (ca) After plating

                *)
                The distance between Top-L1 and Bottom-L2 should be
                as close to 300 um as possible!

                **)
                Select Center Prepreg thickness in order to reach specified
                board thickness
```

Figure 3.1. Reference Design PCB Stack-Up

Refer to the BGM220S data sheet for instructions on how to realize the antenna loop and additional layout design guidelines.

4. Board Size Specific Layout Recommendations

The following sections list additional board size specific layout recommendations on how to maximize the BGM220S antenna efficiency (applicable to both BGM220S12A and BGM220S22A).

Note that some of the listed layout solutions also require tuning the antenna matching, which is described in Section 5. Tuning the Antenna Impedance.

4.1 Optimum PCB Dimensions and Module Placement

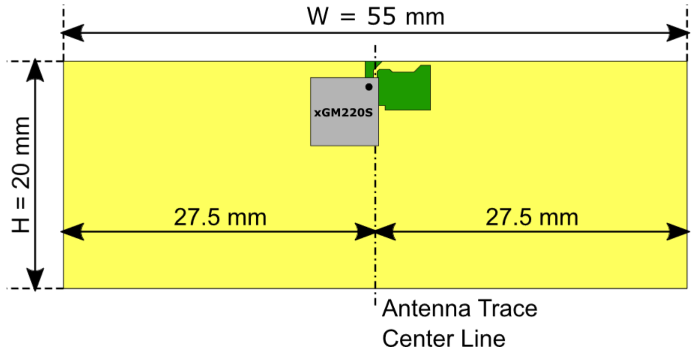

Figure 4.1. Optimum PCB Dimensions and Module Placement

The smallest form factor without compromising antenna performance is as follows:

- Board width (W) is 55 mm
- Board height (H) is 20 mm
- No large metal keep-outs (other than the antenna loop clearance area)

4.2 Recommendations for Small PCB Size

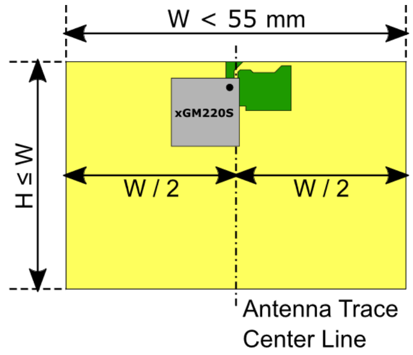

Figure 4.2. Recommended Small PCB Layout – Solid GND Fill

- Degraded antenna gain due to smaller effective antenna area (especially when the board size is smaller than 40 x 20 mm).
- Place the BGM220S and antenna at the center of longest PCB edge.
- If W is between 40 and 50 mm, then H should be 30 mm.
- If W is 40 mm or smaller, then H should be equal to W.

4.3 Recommendations for PCBs with Extended H Dimensions

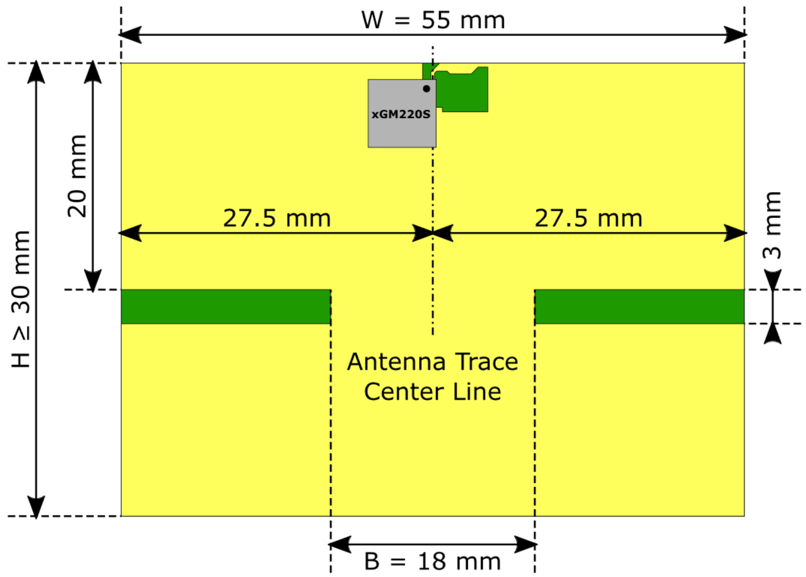

Figure 4.3. Recommended PCB Layout for Extended H Dimension – Horizontal Metal Keep-Outs

Note: Spaces filled in green represent metal keep-out areas.

- Similar antenna performance to Optimum Layout.
- Board width (W) is 55 mm.
- Board height (H) is 30 mm or larger.
- Add 3 mm wide keep-outs as shown on all metal layers to form an 18x3 mm “bridge” between the BGM220S section and the rest of the application board.
- Restrict all signal and power line routing between the BGM220S section and the rest of the application board to the “bridge” between the two areas.

4.4 Recommendations for PCBs with Extended H and W Dimensions (Option #1)

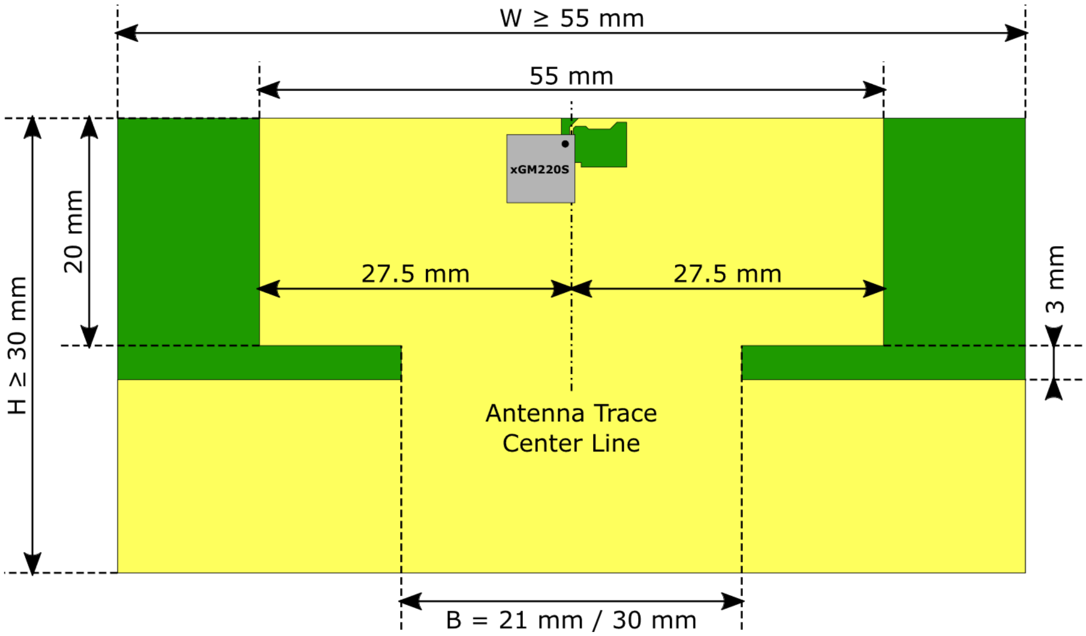

Figure 4.4. Recommended PCB Layout for Extended W & H Dimensions – Wide Metal Keep-Outs

Note: Spaces filled in green represent metal keep-out areas.

- Similar antenna performance to Optimum Layout.
- Board width (W) is 55 mm or larger.
- Board height (H) is 30 mm or larger.
- Wide antenna keep-out areas adjacent to the 20 mm sides of the BGM220S section.
- Add additional 3 mm wide keep-outs as shown on all metal layers to form a B x 3 mm “bridge” between the BGM220S section and the rest of the application board.
- If W is less than 80 mm, then B = 21 mm is recommended.
- If W is 80 mm or greater, then B = 30 mm is recommended.
- Restrict all signal and power line routing between the BGM220S section and the rest of the application board to the “bridge” between the two areas.

4.5 Recommendations for PCBs with Extended H and W Dimensions (Option #2)

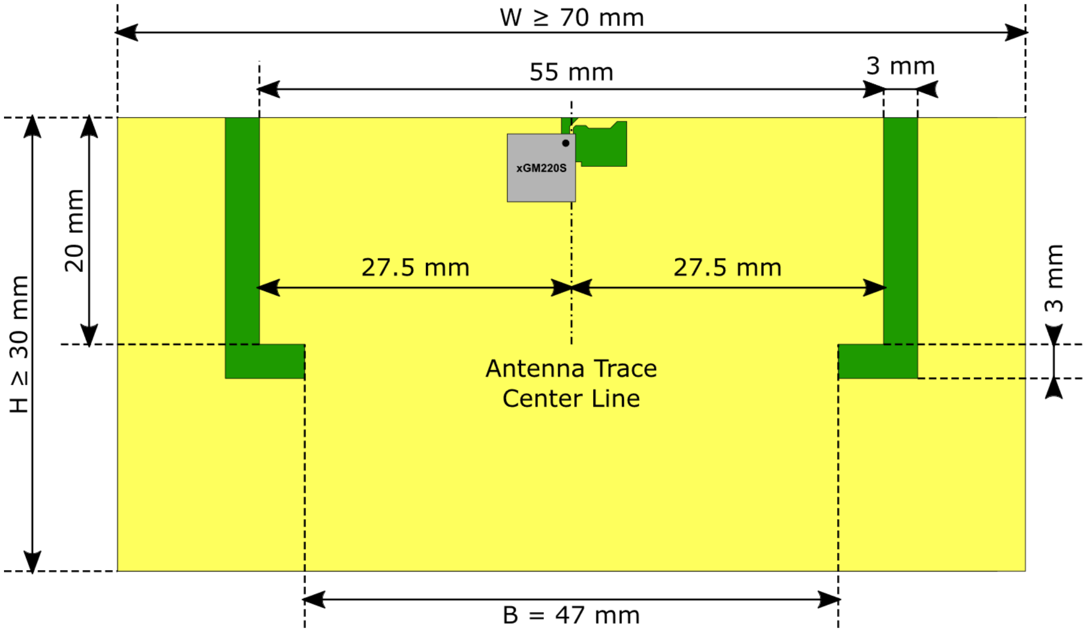

Figure 4.5. Recommended PCB Layout for Extended W & H Dimensions – L-Shaped Metal Keep-Outs

Note: Spaces filled in green represent metal keep-out areas.

- Similar antenna performance to Optimum Layout.
- More space available for application than with the previous Wide Metal Keep-Outs solution.
- Board width (W) is 70 mm or larger.
- Board height (H) is 30 mm or larger.
- Add 3 mm wide L-shaped keep-outs as shown on all metal layers to form a 47x3 mm “bridge” between the BGM220S section and the rest of the application board.
- Restrict all signal and power line routing between the BGM220S section and the rest of the application board to the “bridge” between the two areas.

5. Tuning the Antenna Impedance

WARNING: Any antenna tuning, and/or change of the antenna loop dimensions, is likely to invalidate a modular certification, unless it is done to compensate for the degradation caused by a host board deviating in size or PCB stack-up from the manufacturer's best-case reference. The guidance provided below describes how to address this particular kind of degradation, in which case a permissive change to the modular approval may not be necessary. However, since this is evaluated on a case-by-case basis, please consult your certification house on the best approach.

To achieve optimum realized efficiency, terminate RF_2G4 with a 50 Ω load impedance. If the data sheet design guidelines are followed and the application board size is 55 x 20 mm / 50 x 30 mm, then the antenna input impedance is ~50 Ω by design, meaning ANT_IN can be connected directly to RF_2G4 via a 0R resistor. However, if the design significantly deviates from the data sheet 55 x 20 mm / 50 x 30 mm reference design, then tuning of the antenna impedance may be required.

The antenna matching can be tuned by adjusting only two parameters: W_{loop} and R_TUNE. W_{loop} is the antenna loop width (as shown in Figure 5.1 on page 9) and modifying it primarily affects the antenna resonance frequency. R_TUNE is the value of a series capacitor/inductor between RF_2G4 and ANT_IN (as shown in Figure 5.2 on page 9) and it can be used to cancel out any residual antenna reactance.

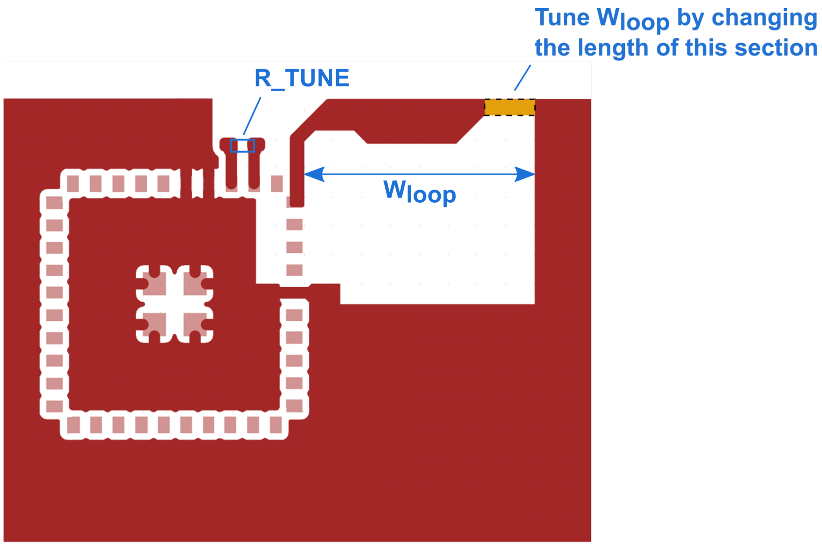

Figure 5.1. R_TUNE and W_{loop}

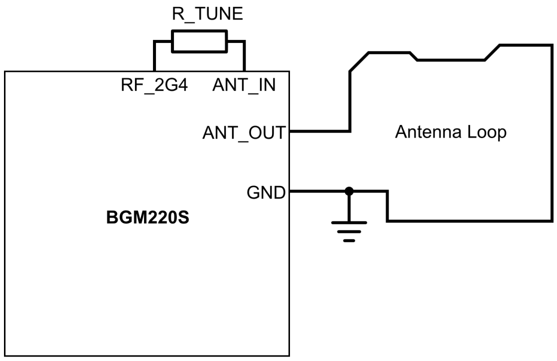

Figure 5.2. Antenna Schematic with Single Tuning Component (R_TUNE)

Recommended W_{loop} and R_TUNE values are summarized in Table 5.1 for BGM220S12A and in Table 5.2 on page 11 for BGM220S22A. Each configuration was tuned assuming 1.6 mm PCB thickness and eps = 4.4 relative PCB permittivity.

5.1 Recommended Tuning Values vs. PCB Size

Table 5.1. Recommended Antenna Tuning Values for BGM220SC12A (based on simulated results, eps = 4.4, PCB Thickness = 1.6 mm)

PCB Width (W) [mm]	PCB Height (H) [mm]	Bridge Width (B) [mm]	W_{loop} [mm]	R_TUNE	Antenna Realized Efficiency Typ. [dBi]	GND Fill Type
20	20	—	6.14	1 pF	-5.8	Solid GND Fill as per Figure 4.1 and Figure 4.2
30	20		5.91	1.4 pF	-2.9	
30	30		5.95	1.4 pF	-2.7	
40	20		5.6	0R	-1.1	
40	30		5.75	5.0 pF	-1.2	
40	40		5.81	3.6 pF	-1.4	
50	30		5.98	0R	-1.0	
55	20				-0.9	
55	40	18	5.77	1.5 nH	-0.8	Horizontal Metal Keep-Outs as per Figure 4.3
55	50				-0.7	
55	60				-0.8	
60	30	21	5.98	0R	-1.5	Wide Metal Keep-Outs as per Figure 4.4
60	40				-1.0	
60	60				-0.8	
70	30				-1.0	
70	40				-1.0	
70	60				-1.0	
80	40	30	5.65	1.7 nH	-1.0	
80	60				-0.8	
100	40				-0.8	
100	60				-0.8	
80	30	47	5.81	0R	-1.0	L-Shaped Metal Keep-Outs as per Figure 4.5
80	40				-1.0	
80	60				-0.9	
100	30				-1.0	
100	40				-1.0	
100	60				-0.9	

Table 5.2. Recommended Antenna Tuning Values for BGM220SC22A (based on simulated results, eps = 4.4, PCB Thickness = 1.6 mm)

<table>
<tr><th>PCB Width (W) [mm]</th><th>PCB Height (H) [mm]</th><th>Bridge Width (B) [mm]</th><th>W_{loop} [mm]</th><th>R_TUNE</th><th>Antenna Realized Efficiency Typ. [dBi]</th><th>GND Fill Type</th></tr>
<tr><td>20</td><td>20</td><td rowspan="8">—</td><td>4.82</td><td>2.7 pF</td><td>-6.6</td><td rowspan="8">Solid GND Fill
as per Figure 4.1 and Figure 4.2</td></tr>
<tr><td>30</td><td>20</td><td>4.7</td><td>5.6 pF</td><td>-3.5</td></tr>
<tr><td>30</td><td>30</td><td>4.75</td><td>5.6 pF</td><td>-3.3</td></tr>
<tr><td>40</td><td>20</td><td>4.6</td><td>1.2 nH</td><td>-1.3</td></tr>
<tr><td>40</td><td>30</td><td>4.72</td><td>0R</td><td>-1.5</td></tr>
<tr><td>40</td><td>40</td><td>4.72</td><td>0R</td><td>-1.7</td></tr>
<tr><td>55</td><td>20</td><td>4.88</td><td>0R</td><td>-1.2</td></tr>
<tr><td>55</td><td>30</td><td>4.88</td><td>0R</td><td>-1.5</td></tr>
<tr><td>55</td><td>40</td><td rowspan="2">18</td><td rowspan="2">4.88</td><td rowspan="2">2.0 nH</td><td>-1.2</td><td rowspan="2">Horizontal Metal Keep-Outs
as per Figure 4.3</td></tr>
<tr><td>55</td><td>60</td><td>-1.0</td></tr>
<tr><td>60</td><td>40</td><td rowspan="3">21</td><td rowspan="3">4.88</td><td rowspan="3">2.0 nH</td><td>-1.1</td><td rowspan="7">Wide Metal Keep-Outs
as per Figure 4.4</td></tr>
<tr><td>60</td><td>60</td><td>-1.1</td></tr>
<tr><td>70</td><td>40</td><td>-1.3</td></tr>
<tr><td>80</td><td>40</td><td rowspan="4">30</td><td rowspan="4">4.66</td><td rowspan="4">2.2 nH</td><td>-1.2</td></tr>
<tr><td>80</td><td>60</td><td>-1.1</td></tr>
<tr><td>100</td><td>40</td><td>-1.0</td></tr>
<tr><td>100</td><td>60</td><td>-1.0</td></tr>
<tr><td>80</td><td>30</td><td rowspan="6">47</td><td rowspan="6">4.74</td><td rowspan="6">1.5 nH</td><td>-1.3</td><td rowspan="6">L-Shaped Metal Keep-Outs
as per Figure 4.5</td></tr>
<tr><td>80</td><td>40</td><td>-1.3</td></tr>
<tr><td>80</td><td>60</td><td>-1.1</td></tr>
<tr><td>100</td><td>30</td><td>-1.2</td></tr>
<tr><td>100</td><td>40</td><td>-1.2</td></tr>
<tr><td>100</td><td>60</td><td>-1.1</td></tr>
</table>

5.2 Tuning the Effects of PCB Relative Permittivity

The PCB relative permittivity (a.k.a. dielectric constant, ε_r) is inversely proportional to the antenna resonance frequency. The relationship between the two is shown in Figure 6.2 for different use cases, however, it can be approximated as:

Increasing/decreasing the PCB relative permittivity by 0.2 decreases/increases the antenna resonance frequency by ~10 MHz

The easiest way of tuning back the antenna resonance frequency is by adjusting the antenna loop width (W_{loop}). As shown in Figure 7.1, the antenna resonance frequency is also inversely proportional to W_{loop} and its effect can be approximated as:

Increasing/decreasing W_{loop} by 0.1 mm decreases/increases the antenna resonance frequency by ~20 MHz

Based on the above information, the W_{loop} offset needed to compensate for the effect of different PCB dielectric constants can be calculated easily. For example, if the PCB dielectric used has epsilon = 4.0, then the W_{loop} values listed in Table 5.1 on page 10 and Table 5.2 on page 11 should be increased by ~0.1 mm.

5.3 Tuning the Effects of PCB Thickness

The PCB thickness has a smaller but more complex effect on the antenna matching. As the total PCB thickness decreases, so does the antenna susceptance (the imaginary part of the complex admittance). However, the resonance frequency also changes slightly (see Figure 6.1 for reference). The magnitude of these detuning effects is slightly different for different configurations (board size, layout epsilon, etc.).

If the PCB thickness is between 0.8 mm and 1.0 mm, then simulation results show that the antenna performance can be tuned back in most cases by adjusting both R_TUNE and W_{loop} as below:

- If Table 5.1 on page 10 and Table 5.2 on page 11 lists an inductor or 0R as R_TUNE for a given configuration, then this R_TUNE value should be increased by 0.6 nH for 1 mm PCB thickness and 1 nH for 0.8 mm PCB thickness.
- If Table 5.1 on page 10 and Table 5.2 on page 11 lists a capacitor for R_TUNE, then its new value (R_TUNE_{new}) should be calculated as: $R\_TUNE_{new} = (R\_TUNE_{orig} \times C) / (R\_TUNE_{orig} + C)$ where C = 7 pF for 1 mm PCB thickness and C = 4.2 pF for 0.8 mm PCB thickness.
- W_{loop} should be also adjusted so that the resonance frequency is increased by ~20 MHz, which (as per Table 5.2 on page 11) translates to ~0.1 mm increase in W_{loop} compared to its nominal value.

5.4 Full Discrete Antenna Matching

Alternative to using W_{loop} and R_TUNE, the antenna impedance may be also tuned by adding a placeholder for a 3-element π-network (as in the figure below) and evaluating the required discrete matching components with a vector network analyzer.

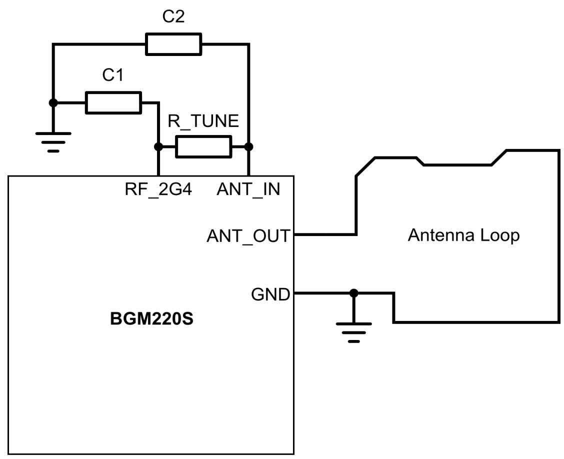

Figure 5.3. Antenna Schematic with 3-element Pi-Match

6. Appendix 1 – Antenna Performance vs. Host PCB Stack-Up

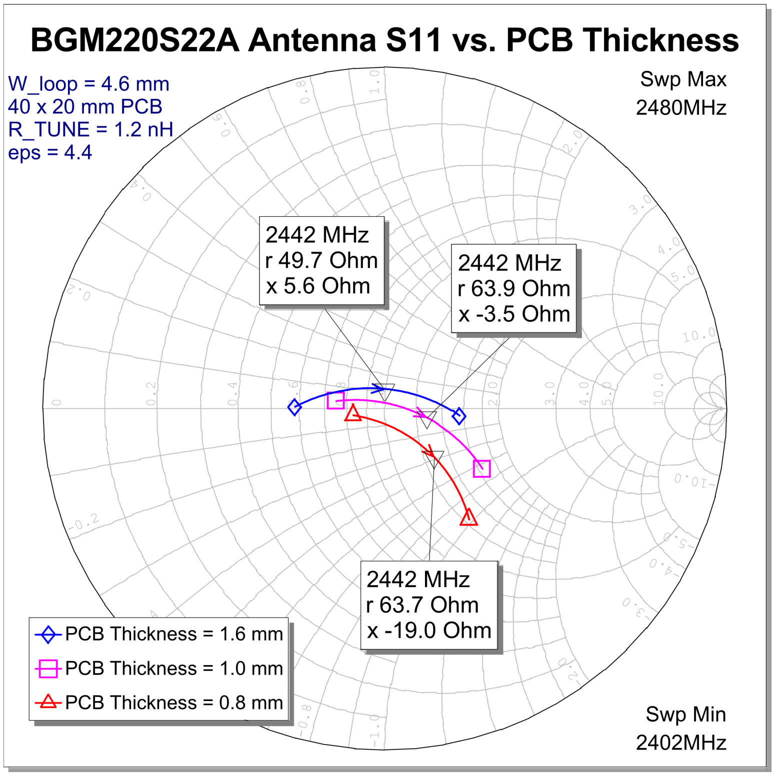

Figure 6.1. BGM220S22A Antenna S11 vs. PCB Thickness (40x20 mm PCB, eps = 4.4)

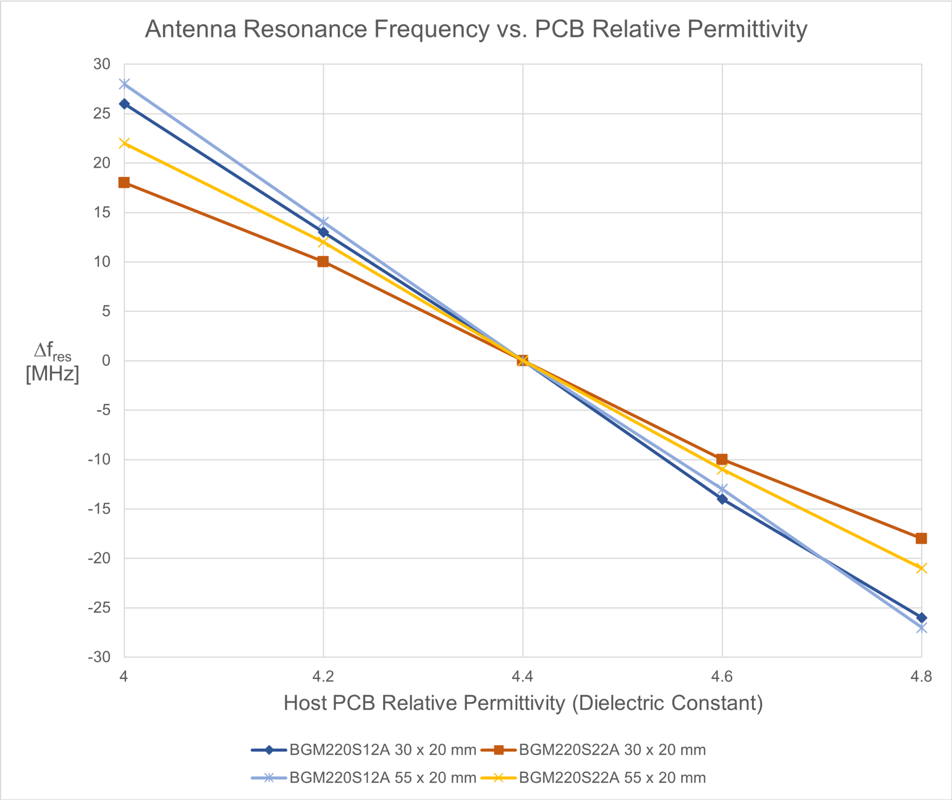

Figure 6.2. Antenna Resonance Frequency vs. Host PCB Relative Permittivity (PCB Thickness = 1.6 mm)

7. Appendix 2 – Antenna Resonance Frequency vs. W_{loop} Size

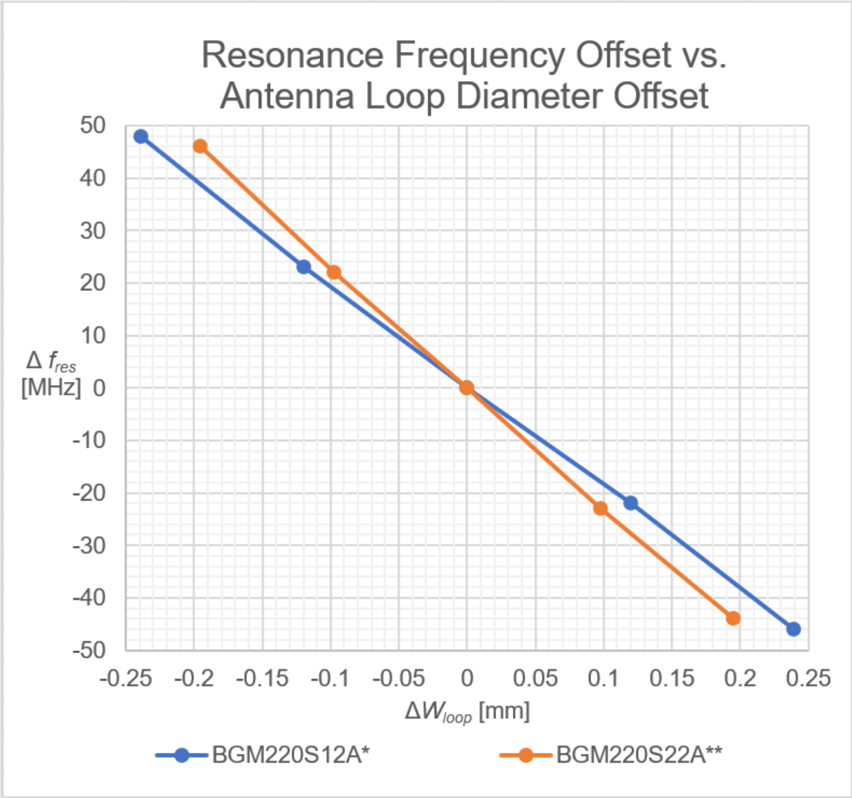

Figure 7.1. Antenna Resonance Frequency vs. W_{loop} Size

Note: *50x30 mm PCB, Total PCB thickness = 1.6 mm, W_{loop} nominal = 5.98 mm

Note: **55x20 mm PCB, Total PCB thickness = 1.6 mm, W_{loop} nominal = 4.88 mm

8. Appendix 3 – Part-to-Part Variation

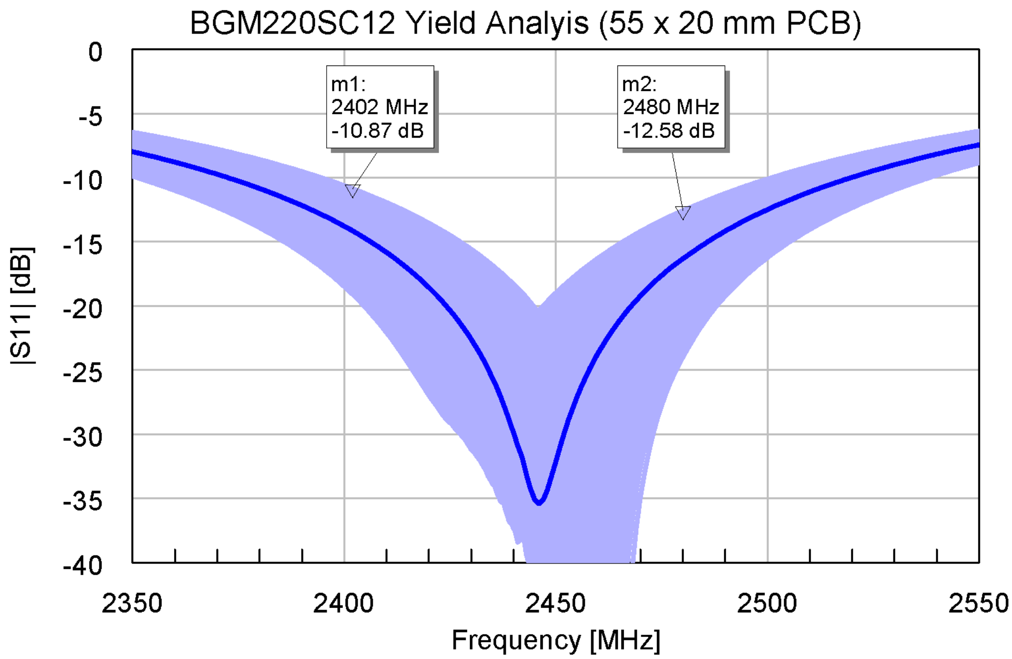

Figure 8.1. Antenna |S11| Yield Analysis (BGM220SC12, 55x20 mm PCB, 1000 samples)

Simplicity Studio

One-click access to MCU and wireless tools, documentation, software, source code libraries & more. Available for Windows, Mac and Linux!

IoT Portfolio	SW/HW	Quality	Support & Community
www.silabs.com/IoT	www.silabs.com/simplicity	www.silabs.com/quality	www.silabs.com/community

Disclaimer

Silicon Labs intends to provide customers with the latest, accurate, and in-depth documentation of all peripherals and modules available for system and software implementers using or intending to use the Silicon Labs products. Characterization data, available modules and peripherals, memory sizes and memory addresses refer to each specific device, and "Typical" parameters provided can and do vary in different applications. Application examples described herein are for illustrative purposes only. Silicon Labs reserves the right to make changes without further notice to the product information, specifications, and descriptions herein, and does not give warranties as to the accuracy or completeness of the included information. Without prior notification, Silicon Labs may update product firmware during the manufacturing process for security or reliability reasons. Such changes will not alter the specifications or the performance of the product. Silicon Labs shall have no liability for the consequences of use of the information supplied in this document. This document does not imply or expressly grant any license to design or fabricate any integrated circuits. The products are not designed or authorized to be used within any FDA Class III devices, applications for which FDA premarket approval is required or Life Support Systems without the specific written consent of Silicon Labs. A "Life Support System" is any product or system intended to support or sustain life and/or health, which, if it fails, can be reasonably expected to result in significant personal injury or death. Silicon Labs products are not designed or authorized for military applications. Silicon Labs products shall under no circumstances be used in weapons of mass destruction including (but not limited to) nuclear, biological or chemical weapons, or missiles capable of delivering such weapons. Silicon Labs disclaims all express and implied warranties and shall not be responsible or liable for any injuries or damages related to use of a Silicon Labs product in such unauthorized applications. **Note: This content may contain offensive terminology that is now obsolete. Silicon Labs is replacing these terms with inclusive language wherever possible. For more information, visit www.silabs.com/about-us/inclusive-lexicon-project**

SILICON LABS

Silicon Laboratories Inc.
400 West Cesar Chavez
Austin, TX 78701
USA

www.silabs.com