antenova

WHITEPAPER

The Complete Guide to Selecting an Antenna To Meet Your Specification

Introduction

Choosing a high-performance wireless module and antenna doesn't guarantee a functional wireless device. The primary cause of underperformance or failure lies in improperly implemented antenna and RF design layouts. Unlike digital components, antenna performance is determined early in the design process. It's crucial to prioritize antenna integration before placing components on the PCB or finalizing the case design. Last-minute antenna additions can lead to serious compromises. Anticipating this is vital for a successful product launch, as oversights may result in network approval failures and significantly impact time to market. This guide covers antenna fundamentals, integrated antenna design and selection, common issues, and measurement for emerging wireless devices.

Antenna Basics

The performance of an antenna is greatly affected by the outer housing and overall system design, for this reason you should never pick your antenna solely based on data sheet comparisons. Architecture and design layout are far more important than just data sheet specs in achieving maximum wireless performance.

> As antennas are effectively transducers, your host PCB and layout plays an equal (if not bigger) part because it will interact substantially with the performance of the entire device.

You need to consider the following when selecting the right antenna(s) to meet your brief:

- What frequencies will you need to support?
- How long is your host PCB? Length of the ground impacts efficiency achieved and is related to wavelength. Sub-1GHz frequencies ideally require 90-100mm of ground plane to work effectively
- How many antennas are required? Consider isolation when using more than one
- Does your system design enable the use of ground plane free, or will it require an on-ground antenna?
- Is there enough room for a SMD (surface mound device) antenna? If not, consider an FPC version stuck to the underside of the product housing
- Will the recommended placement and type of antenna be affected by the use of the product? E.g. the iPhone 4 issue!
- Will use of the product interfere with how the antenna needs to radiate?

Antenna Basics

With embedded antenna, efficiency is probably the most important performance parameter, along with return loss. Efficiency is a measure of what portion of the power supplied is actually radiated by the antenna, after any reflected losses.

Once again, the efficiency of the antenna will be different in every product design to those stated in the data sheet. This is due to design layout, nearby components and the product housing, which absorbs the near-fields of the antenna. Network approval for cellular applications requires a minimum of 40% efficiency.

> A typical standard embedded antenna will range from about 45 to 80%. Achieving an efficiency over 80% is a real challenge, so in an end product application you should be looking at around 50% (+/- 10%)

Ground plane lengths play a big part in determining efficiency, particularly for the sub-1GHz frequencies:

- A 20mm reduction in ground plane length could drop efficiency by 15-20% at sub-1GHz.
- At 1.7-2.2GHz range, the drop will be around 5% or less.

Return Loss (VSWR)

Antenna impedance and the quality of the impedance match (between RF design and wireless module/chip) are characterised by either return loss or Voltage Standing Wave Ratio (VSWR). This impedance parameter measures how much of the power supplied to the antenna reflects back from the antenna terminals. Very good return loss with a compromise on efficiency will still give the same performance.

Isolation

Isolation is a measure of coupling between two or more antennas. In general, they should be placed as far apart from each other as possible, especially if the frequencies are similar. -10dB <1GHz and -20dB for 2GHz of isolation should be the minimum goal.

> “Antennas placed too close together can result in coupling and poor efficiency”

DESIGN TOOL

[A quick guide to 4G and 5G frequency bands](#) →

“”

You cannot simply ‘add in’ an antenna at the last moment as you will face a serious implementation compromise.

Design layout

Even if you select the right antenna and a good wireless module/chip, a poor layout of the RF transmission line (between antenna and wireless module/chip) could result in 50% of more loss in overall performance. This transmission line forms part of the antenna. If spaced evenly along this line, vias can minimise noise and disruption of antenna performance. This strip line should run close to the edge of the host PCB and be as short as possible.

> Antenova has developed a transmission line software for all its SMD antennas to help with this crucial part of your design

Other areas to consider in your design layout are:

- Select an appropriate ground plane length for the frequency antenna you plan to implement.
- The top layer ground has the main effect on antenna performance, compared with other ground layers. Make sure this ground plane is totally filled and adequately spaced vias link this to all other ground layers.
- When using a ground plane free antenna, make sure that all layers under the antenna are free, and no metallic objects like a battery or LCD are placed underneath. • Never run digital tracks underneath the antenna or wires above/below
- Respect the ground plane clearance shown in the data sheet to the side of the antenna.
- Never use narrow tracks or long transmission lines.
- Some antennas need to be placed in the PCB corner, whilst others are to be placed along the long edge of the PCB.

Enclosure

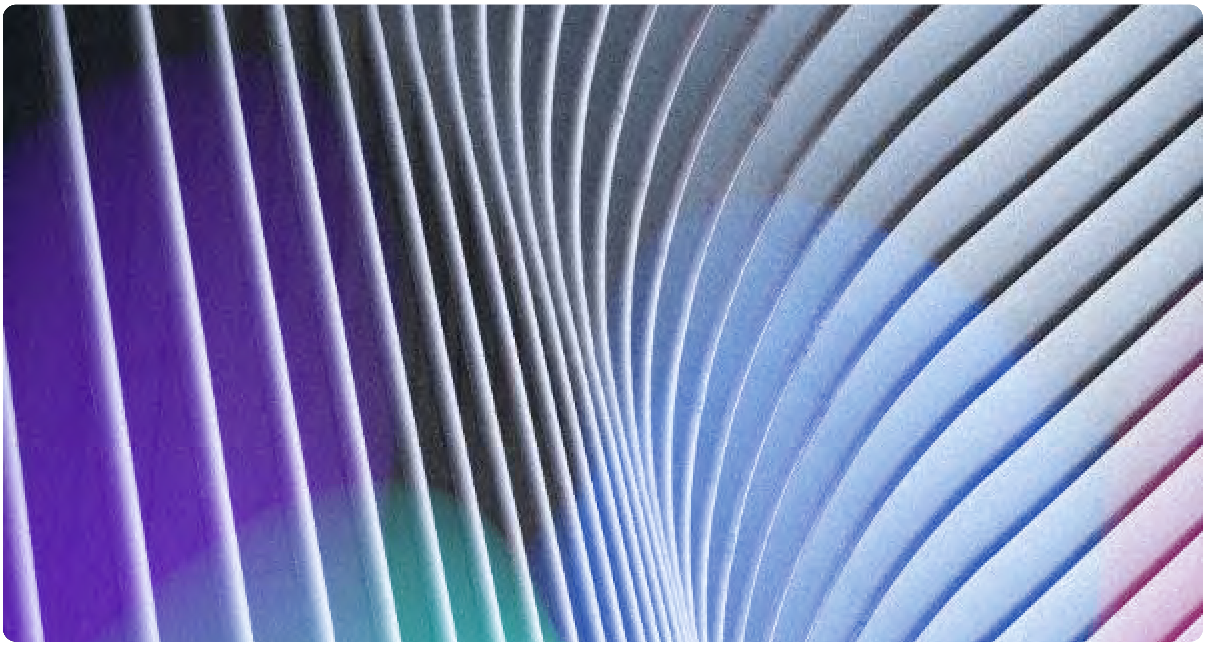
Common plastic materials, like polycarbonate or ABS can absorb RF energy if touching or very close to the antenna. Try to allow a minimum of 3-5 mm gap or greater if possible.

> If a metal enclosure is to be used, then a greater distance and a window in which the antenna can radiate out of may be required. However even with this window, efficiency will be heavily compromised

Also be mindful of LCD's, battery or other metallic objects overhanging the antenna area because this will reduce the radiation window and result in return loss issues.

ARTICLE

4 ways to increase wireless antenna performance in metal devices



Matching/Tuning

Your RF circuit needs to be designed for 50 ohms impedance at the source (wireless module/chip), the transmission line (or co-axial cable) and the antenna. This needs to be close to the antenna to minimise loss in transmission.

Once your design is complete and components are placed near the antenna and the outer housing encapsulates it, it is inevitable that the antenna frequency response will likely shift, resulting in degraded return loss and efficiency. The amount (and direction) of the shift depends on the type of antenna, the overall assembly, but some shifting is inevitable. Some shifting of the frequency response will also occur during normal use when the device is near other objects (product mounted on to metal facia), the user's hand or body etc.

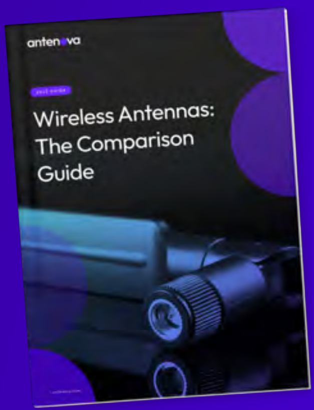
At the feed point of the antenna we highly recommend a "pi" matching circuit consisting of 3 components (inductors and capacitors) for single band antennas and 5 for cellular and LTE antennas. In your final design not all these components may be needed. In which case zero-ohm links can be added.

> “ ”
> This matching circuit allows for the antenna to be optimised for the end application environment, compensate for any frequency shift and to achieve 50 ohms or as close to impedance”

If the end product will always be used in the hand or near the body, then the antenna can be matched and tuned for its working environment as opposed to free space.

Conclusion

The final stages of your design process should not be the time you start considering antenna placement. The performance of your antenna is integral to the operation of your device, and factors encapsulating everything from housing material to PCB size will dictate this performance. Ideally, you need to select and place your antenna onto your device at the earliest opportunity to avoid disaster. This could everything from missed deadlines, or even commercial failure.



Wireless Antennas Compared

You know how important it is to integrate your antenna at the earliest possible opportunity. But what antenna should you integrate? Each antenna has it's own set of pros and cons, and knowing the drawbacks will enable you to intelligently design around these issues. Get our free Wireless Antenna's Compared guide below

Download for free

SHARE KNOWLEDGE WITH RF EXPERTS AROUND THE WORLD

ask.antenova is a global forum for designers and engineers working with wireless technology.

Visit ask.antenova