antenova

WHITEPAPER

A guide to matching circuits, transmission lines and antenna tuning

Contents

Introduction

Designs thrive in environments with the least restrictions and minimal risk of product failure. Part of creating a product is analysing the risk associated with individual components, identifying and selecting the ones with the most desirable functions.

Wireless antennas are not like these other electronic components. Whilst you can predict antenna performance to a certain level, it is highly likely you will face serious compromise if you choose to integrate it at the latter stages of development.

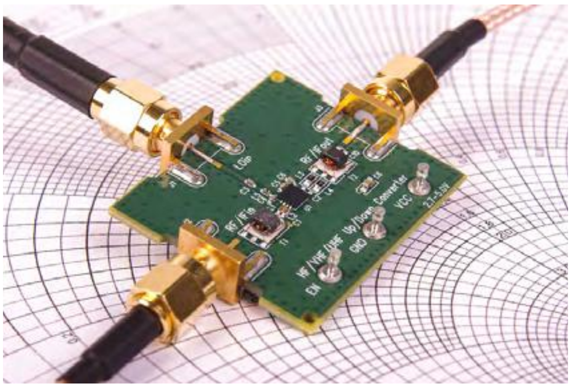
Antenna performance is wholly a product of the operating environment. In every different scenario, the performance of an antenna will vary—be that in free space or whilst embedded within a device. This variation in performance can cause real design headaches for engineers; seemingly design small decisions could prevent an antenna from working.

Matching is a process every device requires to ensure high levels of antenna performance. This guide will take you through the basic principles and best practice to help you avoid a technical development disaster.

What is antenna matching?

One of the most devastating consequences of embedded antenna integration alongside efficiency, is return loss. Objects surrounding an antenna can (and will) interact with radiated electromagnetic fields.

"Antenna Matching is the process of aligning the impedance of both the antenna chip/module and the RF circuitry"



Foreign objects, components and other factors of your design will cause detuning. This means the frequency of an antenna will shift with this interference. These effects occur in all wireless devices, so the cost of using an embedded antenna in this regard are unavoidable. In small, challenging devices these shifts can be more severe. Every device will need adjustment to account for the compact PCB and close proximity to other components and enclosure.

Antenna matching is the process of aligning the impedance of both the antenna chip/module and the RF circuitry. This process allows an antenna to radiate at the intended frequency with minimal deviation, vastly increasing performance capabilities. However, an unmatched antenna will suffer a significantly reduced range and may render the device unable to operate effectively.

The importance of trace lines

Trace lines (or transmission lines) carry received and transmitted signals to and from an antenna. They, therefore, play a key role in the ultimate performance of an antenna. However, unlike typical digital lines, they yield high levels of resistance. The power transfer and return loss of an antenna can be as high as 50% in a poorly matched design.

"Trace lines carry received and transmitted signals to and from an antenna."

Electromagnetic waves are formed of two fields, E, the electric field, and H, the magnetic field. Trace lines operate at 50 Ω, simply due to antennas requiring a ratio of 50:1 to work uninhibited. RF tracks and other elements of the RF system - such as transceivers or power amplifiers—must therefore also match this impedance.

For an embedded antenna, grounded coplanar waveguides (GCPW) are recommended as the antenna can mount on the component side of a PCB and enable improved performance levels. They also remove the need to introduce additional backside manufacturing processes to your PCB.

Measures of antenna mismatches

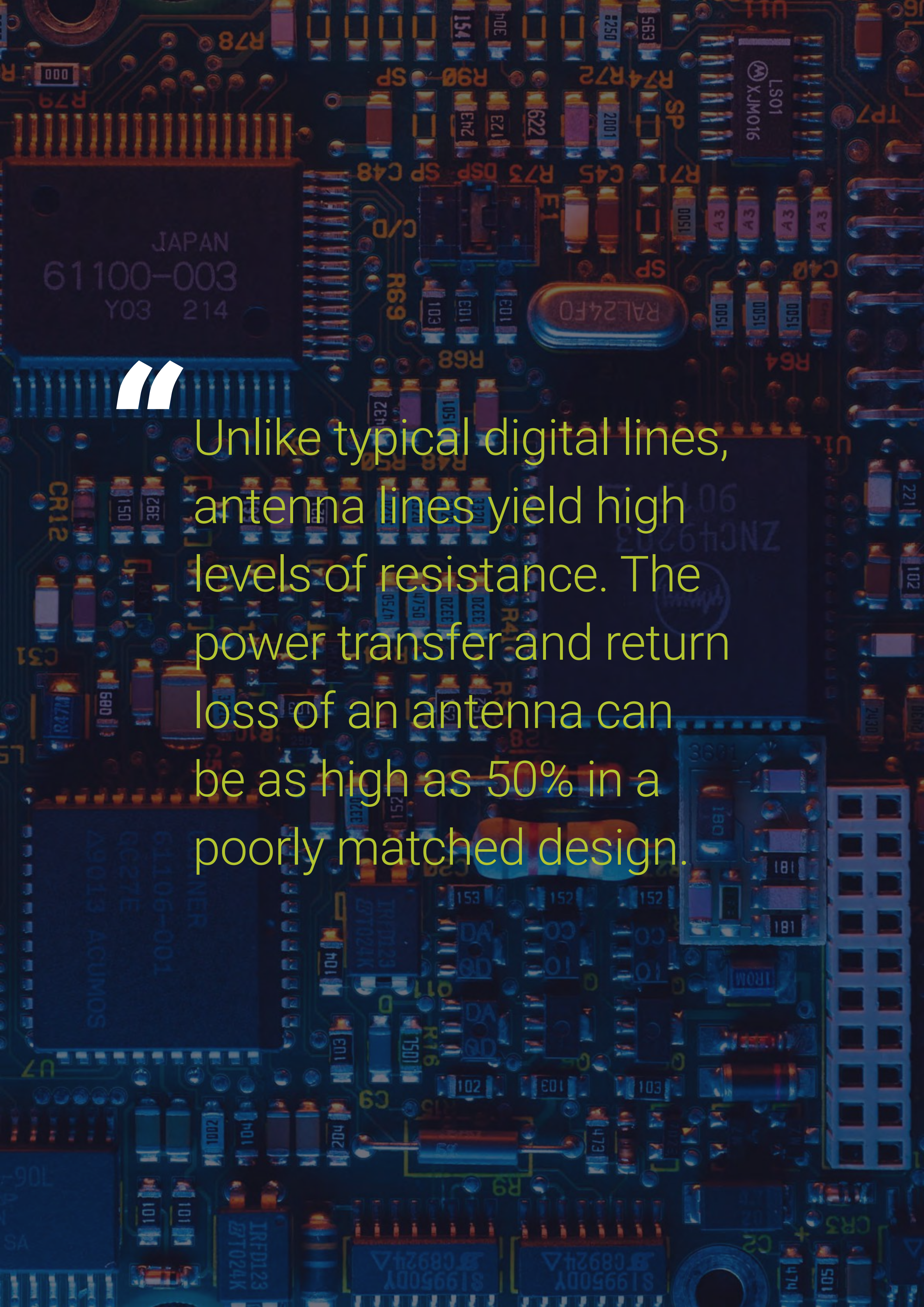
There are several key concepts which shed light on how to correctly match an antenna within a given application.

The VSWR (Voltage Standing Wave Ratio) measures the quality of an impedance match in the form of return loss. Return loss is a measure of reflections caused by a suboptimal impedance match. This measure is one of the most critical parameters in antenna integration, as a well-matched circuit can provide good levels of performance, even when efficiency is inhibited.

"There are several key concepts which shed light on how to correctly match an antenna within a given application."

A lower VSWR is better; the lower the measure, the more energy delivered to the antenna. The presence of a higher VSWR poses issues, as RF energy will be reflected back onto the transmission line—thus do not radiate.

Reflection coefficients measure how much power is reflected from the antenna at the point where the transmission line connects. Ideally, a transmission line



> Unlike typical digital lines, antenna lines yield high levels of resistance. The power transfer and return loss of an antenna can be as high as 50% in a poorly matched design.

would provide 100% of power delivered to the antenna. Although, these levels of performance are highly difficult to achieve in a small device.

Smith charts

Smith Charts plot antenna impedance versus operating frequencies, providing a visual reference for solving impedance mismatches. The lines across the chart are based on multiple equations, providing an illustration of reflection coefficients across various levels of impedance.

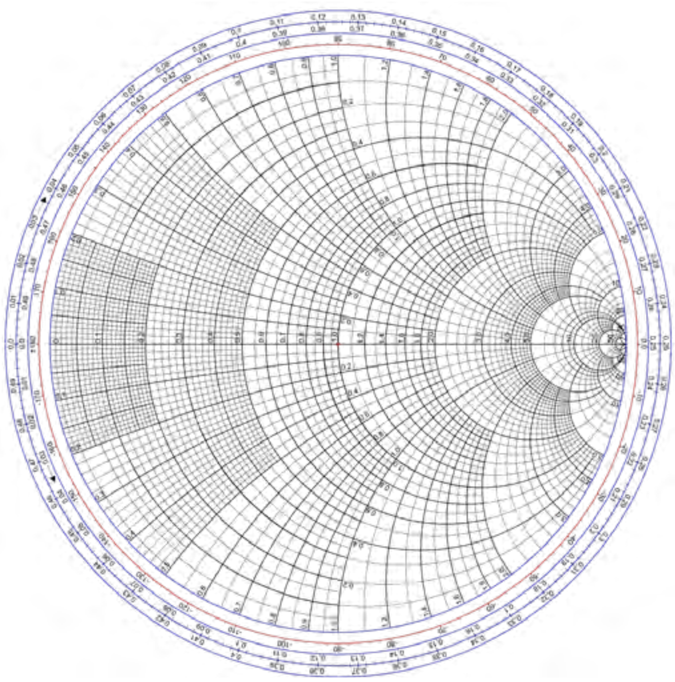


Fig.1: Smith Chart

Historically, these charts were used to speed up the calculation process, although modern day software, such as the Antenova Trace Line Calculator, can solve these sums in microseconds. The centre of the circle represents a perfectly matched antenna, where all power is delivered, whereas the outer ring indicates the maximum reflection coefficient, whereby all power is reflected.

Key matching considerations

There are a number of factors that become important when reducing mismatch losses. The dimensions and length of trace tracks and PCB stack will determine the best design decisions to limit the VSWR.

Trace tracks

Grounded coplanar waveguides (GCPW) are the most appropriate solution for RF tracks in an embedded antenna. Your PCB stack up and dimensions will determine the optimum dimensions for this feed to create the most aligned impedance match possible for your device. This involves calculating the optimum height between the ground plane layer and component layer.

The easiest way to calculate this is by using a calculator, to avoid the mess and lengthy time it spends to learn and apply the formulae yourself. Antenova's trace line calculator allows you to quickly calculate the optimal transmission line dimensions from 3 key parameters—PCB thickness, copper thickness and dielectric constant of the PCB substrate.

PCB material

The associated thickness and dielectric constant of your PCB material (such as FR-4) will play a role in limiting return loss. It's also important to note that any difference in PCB material between development and final manufacture can also have devastating consequences. Engineers often submit their design and find a slight variation in the final manufacture material—be that for cost and efficiency measures—but this shift will mismatch the antenna.

FREE DESIGN TOOL

Get the Antenova Transmission Line Calculator

→

Achieving 50 Ohms impedance

The goal of the matching process is to produce as close to 50 ohms impedance as possible, to minimise return loss (Γ).

Pi matching circuits

The best way to do this for an embedded, surface mount antenna, is by using impedance matching circuit. For this, we recommend a pi (π) matching circuit consisting of three components (inductors and capacitors) for a single-band antenna. Cellular and LTE - multiband - antenna may require 5 components. Pi matching circuits, between the antenna terminal and 50 Ω input ports allow for the antenna resonance to be tuned and optimised. Very often this is for the operating environment rather than free space.

Matching for in-situ performance

If the end product is likely to be used in the hand or near the body (e.g. trackers, wearables), the antenna can be matched and tuned according to the working environment. Pi matching networks become useful to closely control and tune the antenna for these operating environments.

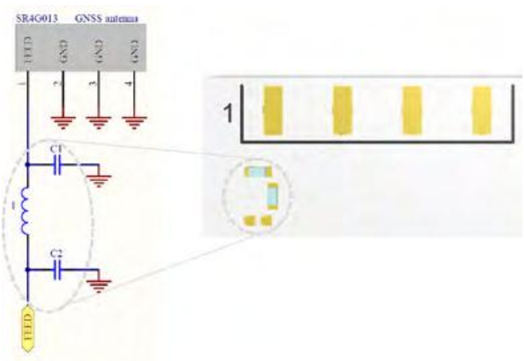


Fig. 2: Pi Matching Network Example for Antenova SR4G013 'Beltii"

"The goal of the matching process is to produce as close to 50 Ohms impedance as possible"

Conclusion

Antenna matching is one of the challenges synonymous with developing a wireless device. Where embedded surface-mount (SMT) antennas are used, often so are Pi matching networks, yet these are often tuned for their operating environments as opposed to free space.

Completing the matching process with minimal experience can be daunting, especially given the implications of every design decision could have on your matching process. Whilst this is important, you will also need to ensure you are following the guidelines of the antenna you are integrating. Without a dedicated RF expert, you may need additional support.