

Antenna Design Guide for the TRF79xxA

Ralph Jacobi and Eddie LaCost

Safety and Security (S2) NFC/RFID Applications

ABSTRACT

This paper describes the design steps for creating and tuning an NFC/high frequency (HF) RFID antenna tuned to 13.56 MHz for the TRF79xxA series of devices. The matching network uses a 50-Ω 3-element match. A 3-element match is recommended as it allows the designer to select the required antenna quality factor (Q) for the application.

Contents

1	Introduction	2
	1.1 Q Factor	2
	1.2 Antenna Coil Placement	2
2	Process for PCB Antenna Design	2
	2.1 Determine the Area and Location for the Antenna Coil.....	2
	2.2 Determine Trace Width and Number of Turns	3
	2.3 Design Tips for PCB Antenna Design.....	4
3	Process for Antenna Tuning.....	5
	3.1 Smith Chart Simulation.....	6
4	Populate the Matching Network.....	9
5	Conclusion	11
6	References	11

List of Figures

1	0-Ω Resistor for Disconnecting the Matching Network	4
2	Smith Chart Capture From Network Analyzer	5
3	Complex Impedance Added to Smith Chart Simulator	6
4	Adding the Parallel Capacitor	7
5	Adding the Parallel Resistor.....	8
6	Adding the Series Capacitor	9
7	Smith Chart Capture of Completed Antenna Match	10
8	SWR Capture of Completed Antenna Match	10

Trademarks

FeliCa is a trademark of Sony Corporation.
All other trademarks are the property of their respective owners.

1 Introduction

When beginning a new antenna design, the targeted application and which NFC/HF RFID technologies will be used for the application has an influence on the decisions made when designing and tuning the antenna.

1.1 Q Factor

The Q factor of the antenna coil is an important consideration to improve read range performance. For applications that use multiple RFID standards (for example, an application that uses both ISO14443A and ISO15693), TI recommends designing for a bandwidth of 1.7 MHz, which results in a theoretical Q factor of 8 based on [Equation 1](#).

$$Q = F_c / BW = 13.56 \text{ MHz} / 1.7 \text{ MHz} = 7.97 \quad (1)$$

A larger bandwidth is required to cover the sidebands of communication for the ISO14443 standard and is especially important to have if using higher over-the-air data rates such as 424 or 848 kbps.

For most applications, a Q factor of 7 to 10 is suitable when the application includes the use of ISO14443A (NFC Type A), ISO14443B (NFC Type B), or FeliCa™ (NFC Type F) technologies.

For an application that uses only the ISO15693 (NFC Type V) technology, the Q factor can be significantly higher, as the sidebands do not need such a wide bandwidth. A Q factor of up to 18 can provide great read range performance and should be considered the upper limit.

1.2 Antenna Coil Placement

When designing an NFC/HF RFID antenna, carefully consider the placement of the coil within the final mechanical encasement and ensure that the antenna coil is kept away from metal. Because NFC/HF RFID is based on magnetic fields, any metal within close proximity to the antenna coil has a dampening effect on the antenna. When the antenna coil is dampened, the inductance changes, and this change negatively affects the tuning of the antenna by causing the center frequency to drift away from 13.56 MHz. Furthermore, the metal may also create eddy currents that affect the received RF signals.

Both issues reduce range performance for the antenna and could even potentially make RFID tags unreadable. However, if it is not possible to avoid placing an antenna coil near the metal case of an enclosure, then a ferrite sheet could be used between the coil and the metal to reduce the effects of the detuning and increase performance of the NFC/HF RFID antenna. If this is the situation, then the antenna must be measured for inductance and tuned when placed within the final enclosure including the ferrite sheet, so that the tuning circuit can account for the changes to the coil inductance caused by the metal and the ferrite sheet.

Recommended spacing for both vertically stacked and side-by-side geometries between metal surfaces and the NFC/HF RFID antenna coil is 10 mm. At that distance, it is likely that metal still has an effect on the antenna coil, such as reducing the Q factor and lowering performance compared to being in free space, but the magnitude of those effects should be reduced to a manageable level.

NOTE: Different types of metal cause varying changes to the dampening and detuning of NFC/HF RFID antennas. This document provides a basic baseline for what to expect, but every application is unique and requires testing and evaluation of the complete system to validate performance requirements.

2 Process for PCB Antenna Design

2.1 Determine the Area and Location for the Antenna Coil

For maximum range, it is usually suggested to use the largest antenna area possible within the constraints of the application, keeping in mind the need to avoid metal as described in [Section 1.2](#). In general, a larger antenna can provide more read range, but there can be diminishing returns as the output power of the TRF79xxA is limited to 200 mW at 5-V operation. If the antenna is too large, then the TRF79xxA cannot properly drive the antenna, and a read hole can occur.

Furthermore, too large of an antenna proportional to the size of an NFC/HF RFID tag also can result in a read hole in which the tag is not detected. Usually read holes are caused because an NFC/HF RFID tag cannot harvest enough passive power from the RF field to properly operate.

When designing a PCB antenna, another consideration is that large PCB planes such as PCB ground planes can also negatively affect the performance of the antenna coil as they act as a metal layer to the coil. Therefore, it must be a priority to avoid laying out the antenna coil above any metal PCB planes. Additionally, approximately 10-mm spacing should be used between the ground plane and the antenna coil to minimize the effect of the ground plane on the inductance of the antenna coil. A smaller distance can be used but may result in reduced performance.

2.2 Determine Trace Width and Number of Turns

When designing the antenna, it is ideal to target for a coil inductance in the range of 1 to 1.5 μH . This is optimal for tuning the antenna to a 13.56-MHz center frequency for HF RFID/NFC.

If the area and shape of the antenna is known, it is possible to calculate the theoretical inductance for different trace widths and numbers of turns. Various online tools can be used to make this calculation.

For example, this website provides a tool that calculates the inductance of a rectangular wire coil: <https://technick.net/tools/inductance-calculator/rectangular-loop/>.

NOTE: For the wire radius, enter the trace width of the PCB traces.

Alternatively, the theoretical inductance for a rectangular wire coil can be manually calculated from Equation 2.

$$L_{rect} \approx N^2 \frac{\mu_0 \mu_r}{\pi} \left[-2(w + h) + 2\sqrt{h^2 + w^2} - h \ln\left(\frac{h + \sqrt{h^2 + w^2}}{w}\right) - w \ln\left(\frac{w + \sqrt{h^2 + w^2}}{h}\right) + h \ln\left(\frac{2h}{a}\right) + w \ln\left(\frac{2w}{a}\right) \right] \quad (2)$$

Where,

- N = Number of turns
- w = Average width of the rectangle
- h = Average height of the rectangle
- a = PCB trace width
- μ_0 = Permeability of the medium

For an FR4 PCB, the relative permeability of the medium can be considered as approximately 1. For thinner PCB substrates, that value could vary and affect the calculation.

The trace width of the antenna should be chosen so that it can be consistently produced by the PCB manufacturer, as variance in the antenna coil changes the inductance and affects the repeatability of the antenna tuning.

Another influence for inductance is the distance between the antenna coil traces. Tight coil windings provide better coupling, and if the coil windings are not tight enough the inductance of the antenna coil can decrease. TI recommends having 0.100-mm or less distance between the antenna coil traces. Keep in mind that manufacturing limitations can influence how tight the coil windings can be.

2.3 Design Tips for PCB Antenna Design

After the dimensions of the antenna are determined, in most cases the next step is to lay out the PCB antenna. In addition to the antenna layout, there are some best practices that are recommended to make the antenna tuning process much easier. These may also apply to external antenna coils that are separate from the primary PCB.

1. Lay out two pads each for the serial and parallel capacitor

This allows for finer adjustments to the capacitors values during antenna tuning, as it is common for the standard value capacitors that are found in lab kits to not provide precise enough changes when trying to optimize the center frequency or bandwidth.

2. Layout the pads for either an SMA or u.FL connector

For PCB antennas, to be able to easily tune and verify the antenna, having pads available to solder on either an SMA or u.FL connector is recommended to provide access to the antenna coil and matching network for a network analyzer tool. The pads should be placed between the 50- Ω point of the TRF79xxA impedance match and the antenna matching network.

3. Use a 0- Ω resistor to isolate the matching network

To isolate the matching network and the rest of the TRF79xxA transmitter matching circuit, a single 0- Ω resistor can be placed between the matching networks (see Figure 1). Place the resistor between the last element of the 50- Ω match and the connector that is used for network analyzer access.

TI strongly recommends this configuration, as the antenna tuning process requires that the network analyzer senses only the antenna coil and the matching circuit. See Figure 1 for an example.

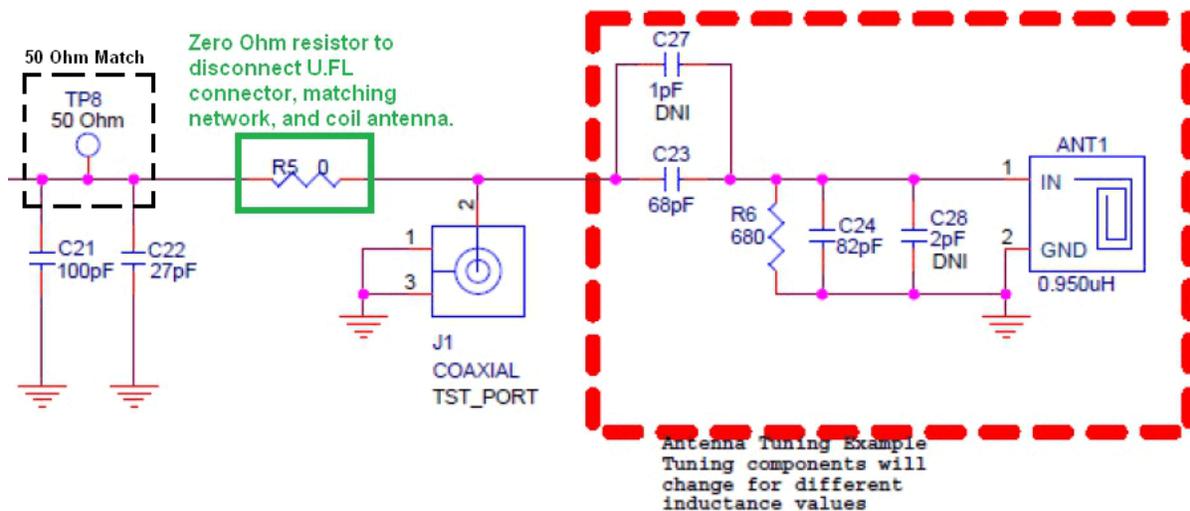


Figure 1. 0- Ω Resistor for Disconnecting the Matching Network

4. If making a prototype, try to use 0603 components for the matching network

This is simply an ease of life suggestion. If a new prototype PCB is being manufactured and the antenna needs to be tuned, the expectation should be that trial and error will be needed to fine tune the antenna. That means a lot of soldering, and it is easier to work with 0603 components than smaller surface mounts such as 0201 components.

3 Process for Antenna Tuning

When tuning an antenna for HF RFID applications, the first step is to measure the inductance and complex impedance of the antenna coil. A network analyzer spanning 12 MHz to 15 MHz should be used for this measurement. Ensure that the analyzer has been properly calibrated. If using a long cable between the antenna and the 50-Ω point, TI recommends calibrating the network analyzer with the cable attached to calibrate out any characteristics of the cable.

A Smith Chart reading is the first measurement needed when using the network analyzer. This should provide the coil inductance and the values for the real and imaginary components of the coil (see Figure 2).

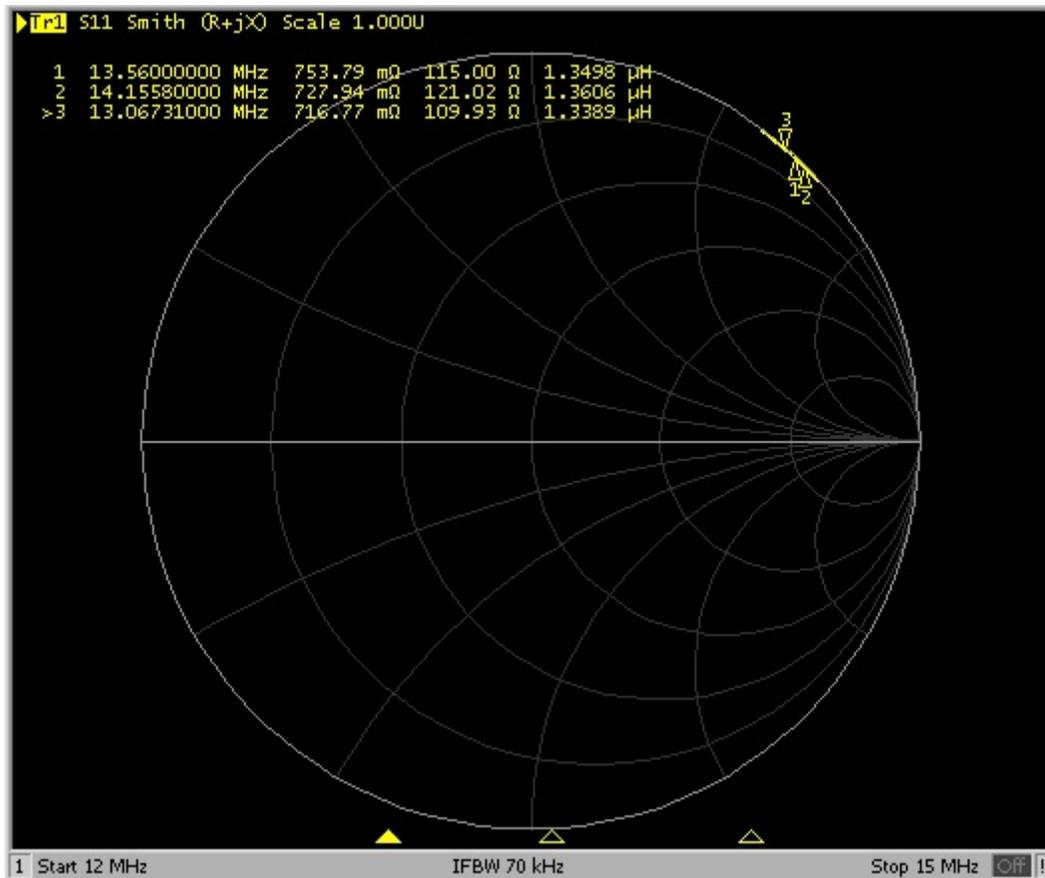


Figure 2. Smith Chart Capture From Network Analyzer

For the example in Figure 2, the inductance is at approximately 1.35 μH, which is within the previously mentioned ideal range for HF RFID antenna tuning. The real component is approximately 754 mΩ, and the imaginary component is approximately 115 Ω. These values are needed to calculate initial values for the antenna matching network.

3.1 Smith Chart Simulation

Now that the inductance and complex impedance are known, the next step is to simulate tuning with Smith Chart simulation software. The following example uses the Smith V3.1 software made by Fritz Dellsperger (<http://fritz.dellsperger.net/smith.html>).

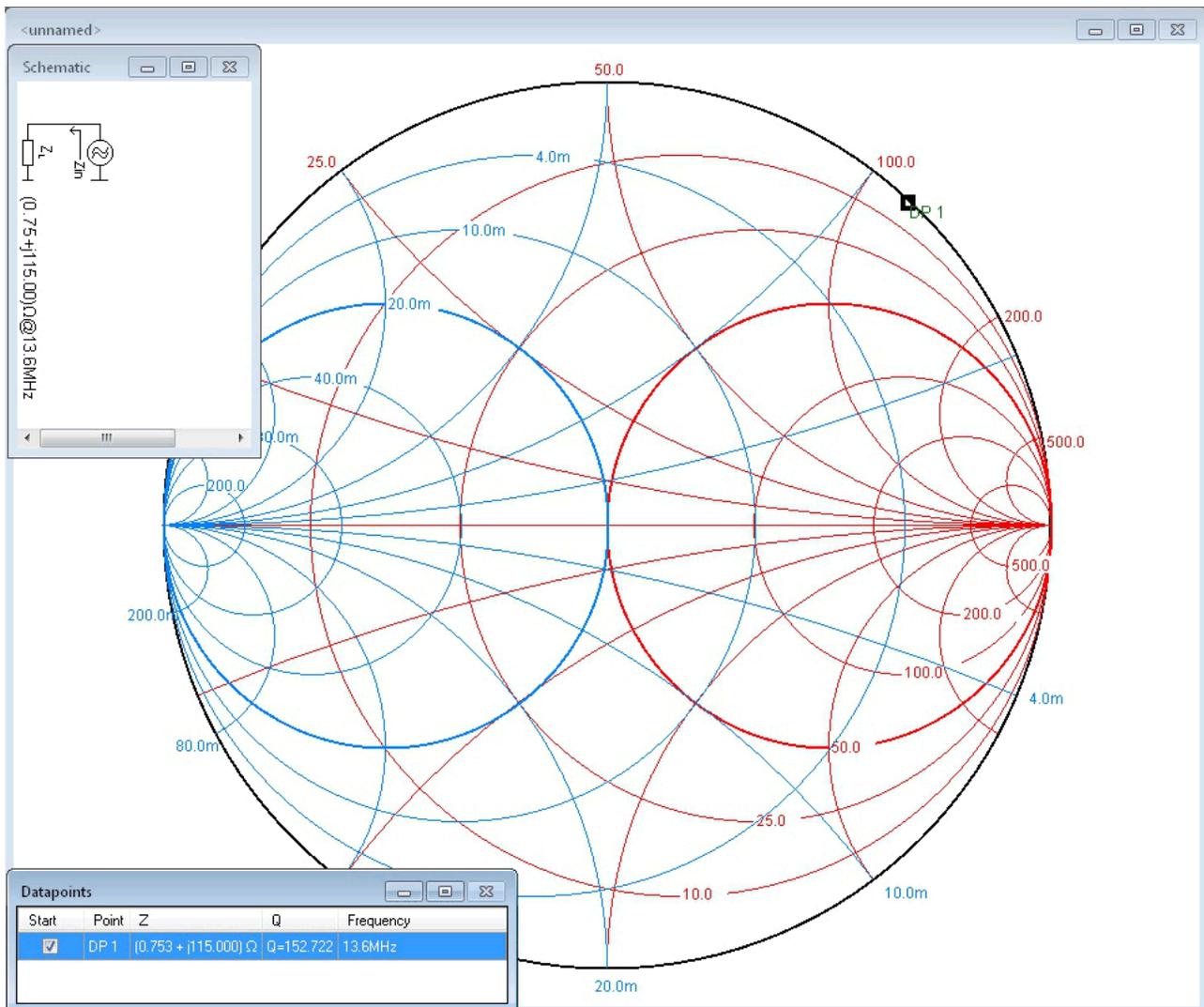


Figure 3. Complex Impedance Added to Smith Chart Simulator

The first step is to add the real and imaginary component values measured from the Smith Chart into the simulator. Be sure to select the frequency to be 13.56 MHz as well. These values provide an initial data point that should be in line with the Smith Chart capture as seen in [Figure 3](#).

To begin placing components on the matching network, the desired Q factor should be used to estimate the value of the parallel resistor. This calculation can be done with [Equation 3](#).

$$R_p = jX \times Q$$

where

- jX is the imaginary component
- R_p is the parallel resistor value

(3)

This example tunes an antenna for a TI EVM that supports both ISO14443A and ISO15693 systems, so the targeted Q is 7 to 9.

When laying out components on the Smith Simulator, the recommended order is: parallel capacitor, parallel resistor, and series capacitor.

Placing the parallel capacitor should bring the data point near the hemisphere at 200.0 (see [Figure 4](#)).

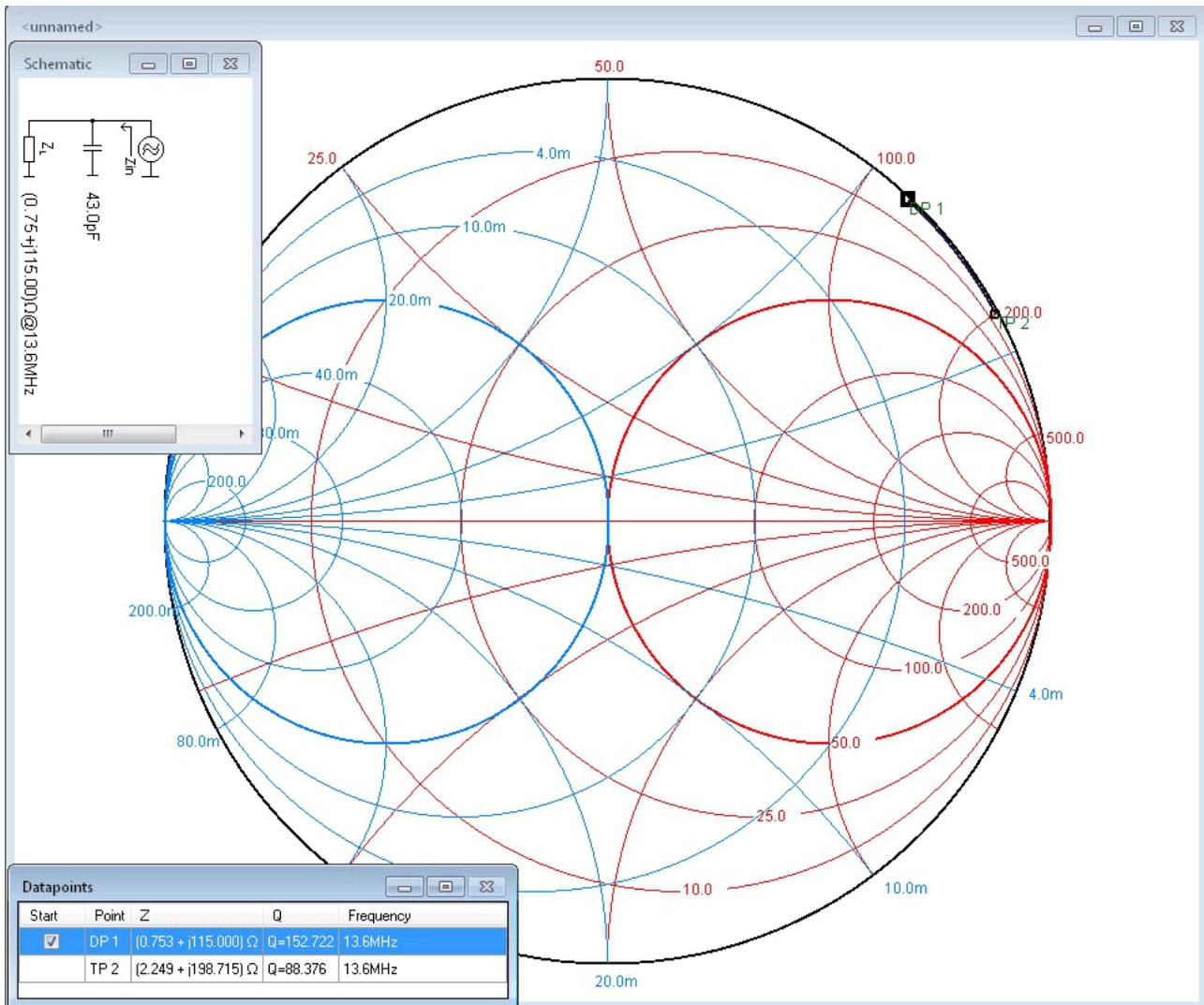


Figure 4. Adding the Parallel Capacitor

Now the goal is to bring the data point toward the bolded red circle by adding the parallel resistor (see [Figure 5](#)). If there is a targeted Q, then calculate the parallel resistance to achieve that goal and use that value in the Smith Chart simulator.

Adjustments to the parallel capacitor value can bring the data point closer to the bolded red circle if needed. However, as this is just a simulation tool and results differ slightly on physical boards, such adjustments are not required.

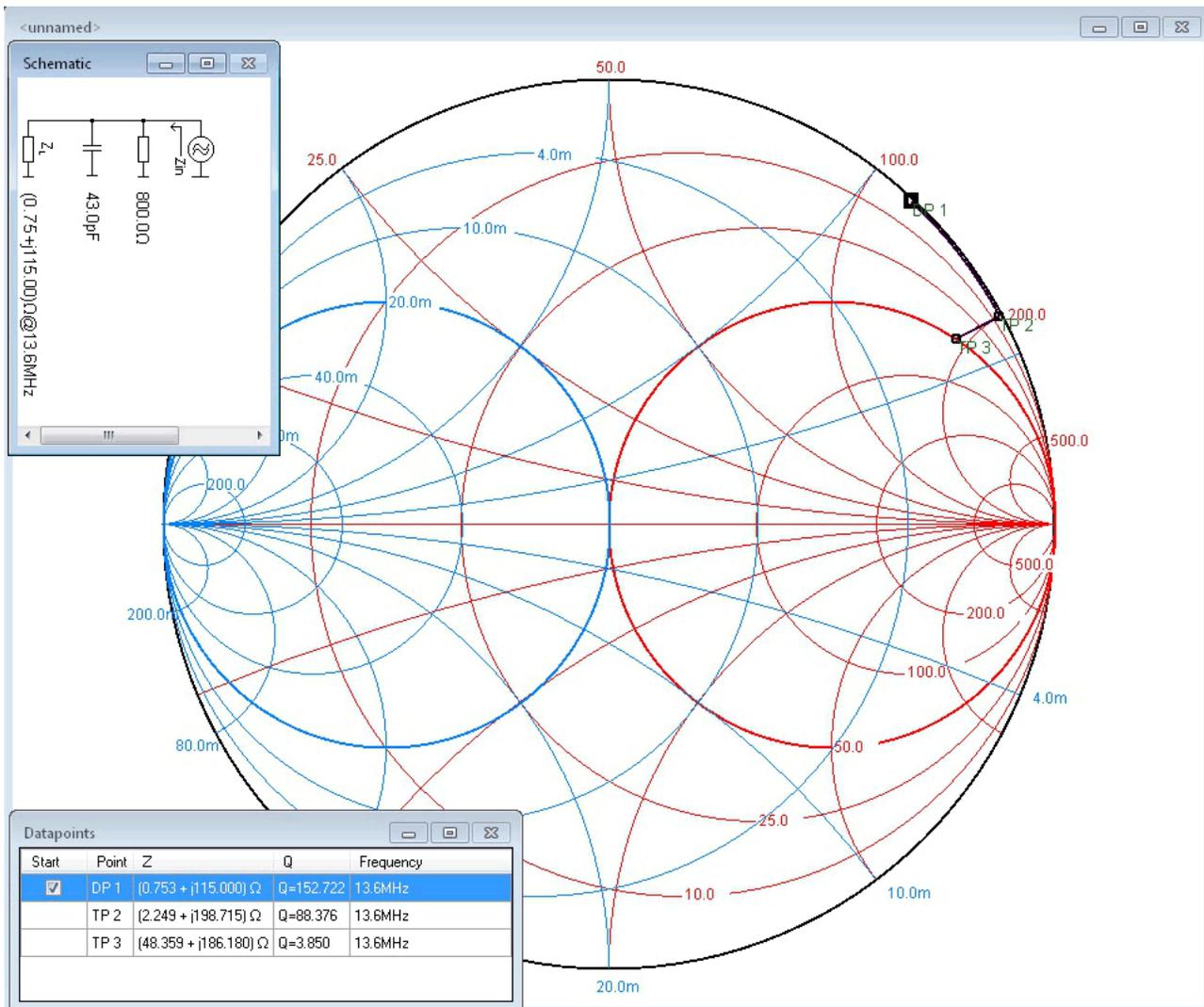


Figure 5. Adding the Parallel Resistor

Placing the resistor on the bolded red circle resulted in a value of 800 Ω (see Figure 5), which results in a Q of 7 (see Equation 4).

$$R_p = jX \times Q \rightarrow R_p / jX = Q \rightarrow 800 / 115 = 6.96 \tag{4}$$

This value is on target for the desired application requirements, so no adjustments need to be made to either component on the simulation.

With the last component placed, the data point is now nearly centered on the 50-Ω point (see Figure 6) of the Smith Chart. The final values used are Cp = 43 pF, Rp = 800 Ω, and Cs = 63 pF.

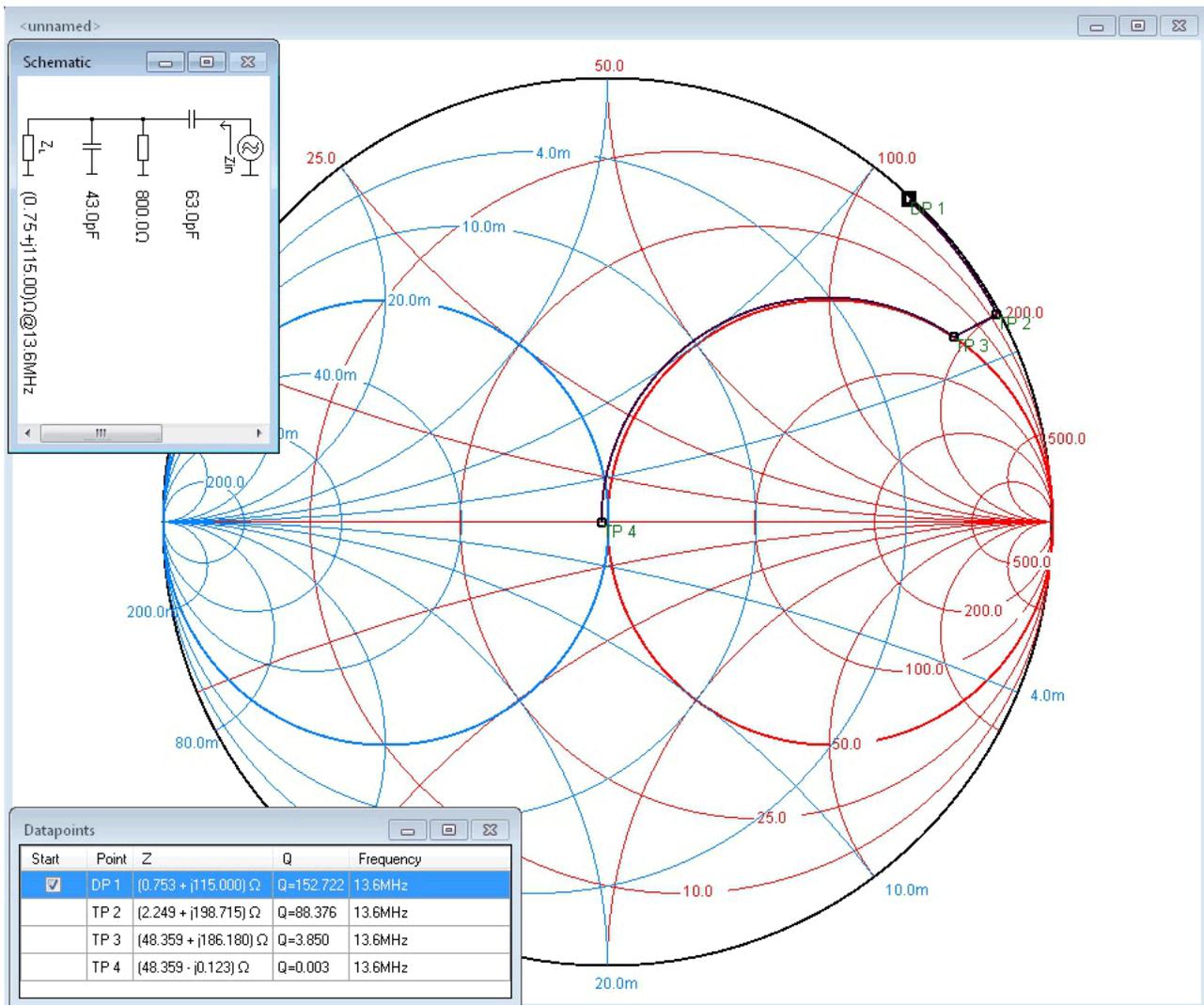


Figure 6. Adding the Series Capacitor

4 Populate the Matching Network

Now with the theoretical match completed, the next step is to populate the chosen tuning components and connect the tuned antenna to the network analyzer to look at the Smith Chart and SWR plots of the resulting circuit.

This step is important as Smith Chart simulations are done in ideal circumstances with no variance in component values and no accounting for the topology of the antenna. In the case of this example, the initial components selected from the software were not sufficient to meet the design requirements of 13.56 MHz with a Q factor of 7 to 9 as targeted.

The final components actually used were $C_p = 47 \text{ pF}$, $R_p = 700 \Omega$, and $C_s = 68 \text{ pF}$. These values resulted in a center point right at 13.56 MHz and brought the Q factor to approximately 9 (see Figure 7). The simulation proved useful to get the values close to target, but it did not account for component variability or any effects that the materials, layout, or manufacturing variances can introduce.

Regarding component variability, TI strongly recommends using 1% tolerance components for the 50- Ω matching network to provide consistent and repeatable results for mass production boards.

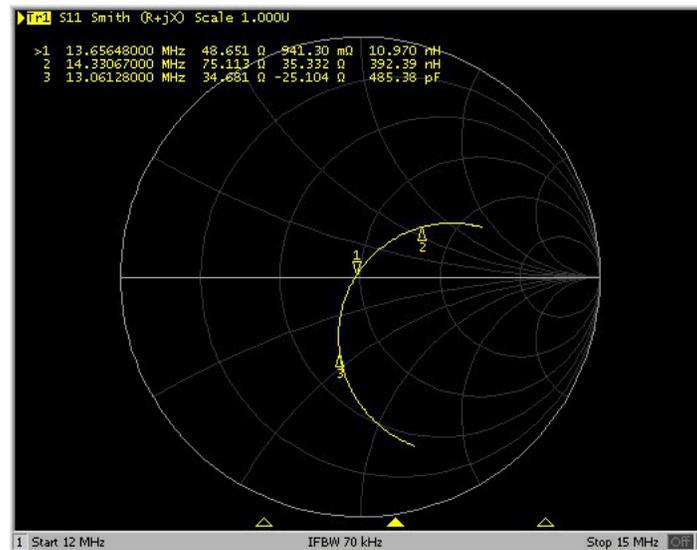


Figure 7. Smith Chart Capture of Completed Antenna Match

The center frequency in Figure 7 is 13.56 MHz. It is not very common for a single set of standard-value capacitors to land the final center frequency on 13.56 MHz, so if precision is desired, often the capacitance needs adjustments of 1 to 5 pF to exactly match 13.56 MHz. This adjustment is the reason for the recommendation for two pairs of pads for the capacitors in Section 2.3. Precise tuning is most important when creating a high-Q antenna for ISO15693, as the sidebands are cut off if the bandwidth is too narrow and the center frequency is off tuned, and that cutoff results in poor performance.

The bandwidth and Q factor can be determined from the SWR plot on a network analyzer. The bandwidth is defined as the difference between the 2-dB points of the SWR plot.

As seen on the SWR capture (see Figure 8), the 2-dB points are at 14.41 MHz and 12.85 MHz, resulting in a span of 1.56 MHz total. The Q factor is F_c/BW , so 13.56 MHz / 1.56 MHz results in 8.69. That Q factor is within the targeted range of 7 to 9 for this example application.

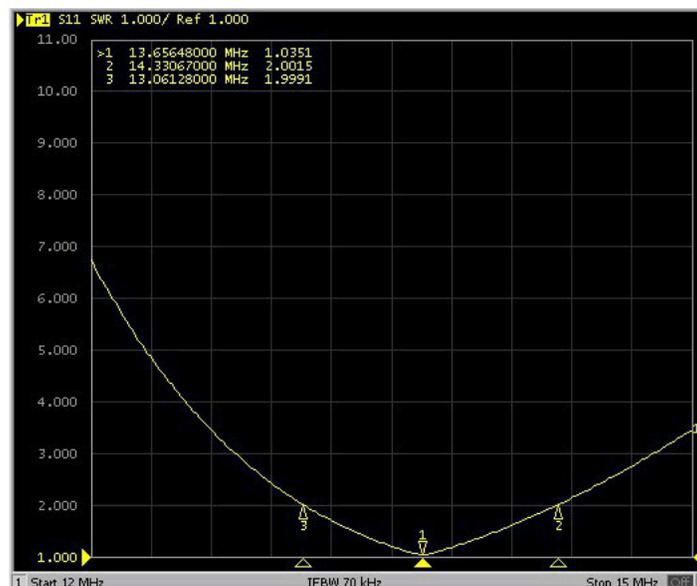


Figure 8. SWR Capture of Completed Antenna Match

5 Conclusion

It is important for NFC and HF RFID systems to have properly tuned antennas to ensure robust performance and optimal read ranges. The process for tuning an antenna coil to 13.56 MHz and optimizing performance is very procedural. Following the steps of this guide including best PCB layout practices makes the process straightforward. Using 1% tolerance components for the tuning component values derived from the antenna tuning process helps provide consistent and repeatable level of performance across mass production boards.

6 References

1. [TRF7970A Multiprotocol Fully Integrated 13.56-MHz RFID and NFC Transceiver IC](#)
2. [TRF7960A Multiprotocol Fully Integrated 13.56-MHz RFID Reader/Writer IC](#)

Revision History

NOTE: Page numbers for previous revisions may differ from page numbers in the current version.

Changes from February 24, 2018 to March 10, 2020 (from B Revision (Feb 2018) to C Revision)	Page
• Changed Figure 2	5

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES "AS IS" AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, or other requirements. These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI's products are provided subject to TI's Terms of Sale (www.ti.com/legal/termsofsale.html) or other applicable terms available either on ti.com or provided in conjunction with such TI products. TI's provision of these resources does not expand or otherwise alter TI's applicable warranties or warranty disclaimers for TI products.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265
Copyright © 2020, Texas Instruments Incorporated