Introduction

This document provides basic considerations for the design of boards featuring the ST25R3916 device, to mitigate unwanted emissions and to keep the overall noise floor to a low level.

The interference between ST25R3916 and other active components (like receipt printers or displays) must be taken into account during the layout phase, to assure clean signal transmission.

Particular attention must be paid to signals propagating along traces, and to the spectrum content of each of them. Signals have to be analyzed in the time and frequency domains to track unwanted emissions back to the source, and take counter measures.

With a proper PCB layout, many EMI problems can be minimized, enabling the application to comply with FCC, RED and/or CISPR standards.
List of figures

Figure 1. Square wave ................................. 6
Figure 2. Spectrum of a square wave ....................... 6
Figure 3. Trapezoidal wave ................................ 6
Figure 4. Low-pass filtering ................................ 7
Figure 5. Position of the low-pass filter on the PCB ............. 8
Figure 6. ST25R3916 with symmetrical matching layout .......... 9
Figure 7. Loop areas on a PCB ............................... 10
Figure 8. Routing of matching network ....................... 10
Figure 9. Avoiding slots in the GND plane ..................... 12
Figure 10. Common mode filter - Position on the PCB ............. 13
Figure 11. Common mode filter - Schematic .................... 13
Figure 12. Position of the decoupling capacitors (top view) ....... 16
Figure 13. Position of the decoupling capacitors (bottom view) ... 17
Figure 14. Schematic of the decoupling capacitors ............... 18
Figure 15. Detail of power supply schematic ................... 19
Figure 16. ST25R3916 thermal pad ......................... 20
Figure 17. Vertical separation between analog and digital signals 22
Figure 18. 4-layer PCB configuration .......................... 22
# Acronyms

<table>
<thead>
<tr>
<th>Acronym</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>AGND</td>
<td>Analog GND</td>
</tr>
<tr>
<td>DGND</td>
<td>Digital GND</td>
</tr>
<tr>
<td>EMC</td>
<td>Electro magnetic compatibility</td>
</tr>
<tr>
<td>EMI</td>
<td>Electro magnetic interference</td>
</tr>
<tr>
<td>EUT</td>
<td>Emitter under test</td>
</tr>
<tr>
<td>ESD</td>
<td>Electro static discharge</td>
</tr>
<tr>
<td>GND</td>
<td>Ground (reference) level for voltages</td>
</tr>
<tr>
<td>IC</td>
<td>Integrated circuit</td>
</tr>
<tr>
<td>PCB</td>
<td>Printed circuit board</td>
</tr>
<tr>
<td>RFI</td>
<td>Radio frequency input</td>
</tr>
<tr>
<td>RFO</td>
<td>Radio frequency output</td>
</tr>
<tr>
<td>USB</td>
<td>Universal serial bus</td>
</tr>
</tbody>
</table>
2 General recommendations

In PCB design there are guidelines that help optimize EMC performance:

- Always consider and determine where and how the return currents are flowing.
- Do not route signals over ground gaps.
- Partition mixed-signal PCBs with separate analog and digital sections.
- Do not split the current return plane; use one solid plane under both analog and digital sections of the board.
- Route digital signals only in the digital section of the board (for all digital related layers).
- Route analog signals only in the analog section of the board (for all analog related layers).
- In case ground or power planes are split for a specific reason (i.e. mechanical and or electrical), do not run any traces across the split on an adjacent layer.
- Traces (analog or digital) that must go over a power plane split must be on a layer adjacent to a solid ground plane (analog or digital).
- A/D and D/A converters, as well as most other mixed-signal ICs, should be considered as analog devices with a digital section, not digital devices with an analog section.
- The AGND and DGND designation on the pins of a mixed signal IC refers to where the pins are connected internally, and it does not imply where or how they should be connected externally. On most mixed-signal ICs, both the AGND and DGND pins should be connected to the analog return plane.
- The digital decoupling capacitor should be connected directly to the digital ground pin.
- The decoupling capacitors are needed to supply, through a low-inductance path, some or all of the transient power supply current required when an IC logic gate switches.
- Decoupling capacitors are needed to short out, or at least reduce the noise injected back into the power ground system.
- Decoupling is not the process of placing a capacitor adjacent to an IC to supply the transient switching current; rather it is the process of placing an L-C network adjacent to the IC to supply the transient switching current.
- The value of the decoupling capacitor(s) is important for the low-frequency decoupling effectiveness.
- The value of the decoupling capacitor(s) is not important at high frequencies. At high frequencies, the most important criteria is to reduce the inductance in series with the decoupling capacitors.
- Effective high-frequency decoupling requires the use of a large number of capacitors.
- Place decoupling capacitors as close as possible to the device.
- Route RFI and RFO signals symmetrically, and avoid long signal traces for the matching network. Keep the traces between RFO1 and RFO2 close to each other, and do the same for RFI1 and RFI2.
- The matching components need be placed close to each other, and symmetrically.
3 Basic considerations

Most of the signals generated by ICs are in the form of periodic square wave signals (Figure 1), with a spectrum similar to the one shown in Figure 2.

Figure 1. Square wave

![Square wave diagram]

Figure 2. Spectrum of a square wave

![Spectrum of a square wave diagram]

As an ideal square wave cannot be generated in real devices, we have to deal with trapezoidal pulses from ICs or clocks, with finite slew rise and fall times (Figure 3). The amplitude of the higher frequency harmonics depends on the rise and fall times of the signal, dropping with longer rise times.

Figure 3. Trapezoidal wave

![Trapezoidal wave diagram]
For the ST25R3916, the slew rate is internally controlled. Another method to reduce harmonics is to use low-pass filtering to round the edges. This is accomplished by an LC combination at the output of the RF drivers, as shown in Figure 4.

Figure 4. Low-pass filtering
It is important to position the filter network as close as possible to the output stages (as in Figure 5), to avoid unwanted radiation over long traces.

Figure 5. Position of the low-pass filter on the PCB
4 Radiation from traces

One of the sources of radiation can be identified in the exposed traces. To understand how traces radiate, the signals must be analyzed during their propagation. A signal propagating along traces can be divided into two main parts, namely a differential signal and a common mode signal.

4.1 Differential signals

The total radiated field from a differential signal propagating along two traces tends to cancel itself, and in ideal conditions is zero. The same effect applies to the output drivers of ST25R3916, which are operating in differential mode. However, due to component tolerances and signal asymmetries along the traces, differences in the current flow may appear and consequently a non “quasi-zero” electric field is generated. Therefore, take care to layout the matching topology in a symmetrical way.

In addition, to control differential-mode radiated emission, it is important to minimize the loop areas (see Figure 7) formed by signal traces and their return current paths. Especially for clocks signals, the length needs to be minimized, to form the smallest possible loop areas.
4.1.1 RFO and RFI routing

The transmit and receive stages of the ST25R3916 are differential pairs and need to be treated carefully during the PCB layout to minimize unwanted signal coupling and radiated emissions. These signals should be routed in an internal layer, but to avoid a large number of vias interconnecting the traces with the matching components, the complete matching network is placed on one side (the top layer) of the PCB. The return current will flow under the traces in the GND layer. It is important to have no cuts in the GND layer below the matching circuit.
It is essential to avoid so-called through-vias in the matching layout, because they only contribute to unwanted emissions. Note how matching components are close to each other to reduce the trace length from RFO to the antenna feed. It is not advised to have long signal traces between the LC filter and the remaining matching components. The inductors after the RFO are positioned perpendicularly to each other to minimize coupling effects.

The RFI lines are routed symmetrically, at a reasonable distance from the RFO lines. The receive signals are decoupled from the middle point of the matching network and crossing the series capacitance where the carrier signal is relatively low. Never route the RFI signals separated from each other, and do not use different signal lengths.

Alternatively, the RFI lines can be routed through the middle of the matching network back to the RFI pins.

Additionally, all the components (capacitors) to GND along the RFO path have been connected to GND by using multiple vias to minimize inductance, and consequently avoid unwanted resonances.
4.2 **Common mode signals**

Common mode radiation mainly emanates from the connected cables of a PCB system.

The PCB reference ground plane inductance (and consequently the ground voltage) is the major contributor to the common mode radiation. Common mode radiation is more effectively produced than differential mode radiation, even a small amount of common mode current can result in significant emission problems.

To reduce common mode emissions
- keep the current supply path close to the current return path
- reduce cable length
- minimize the common-mode voltage (usually the ground potential). Reducing the ground impedance can be done by using solid ground planes or ground grids, and by avoiding slots in the ground plane (see the differences in *Figure 9*).

![Figure 9. Avoiding slots in the GND plane](image)

Take care of connected cables (if any), and shield them properly. This can be done, for instance, using common-mode impedance choke in series with the cable, with some isolation between connections from the enclosed cable to the PCB ground.

An integrated common mode filter (ECMF02-4CMX8) for USB D+ and D- lines and ESD protection for all lines can be used, as exemplified in *Figure 10* and *Figure 11*, showing, respectively, the position on the PCB and the schematic.
Figure 10. Common mode filter - Position on the PCB

Figure 11. Common mode filter - Schematic
4.2.1 SPI data signal routing

The SPI data signals from the ST25R3916 to the MCU must be routed, as much as possible, with equal length and controlled impedance.

Keep the trace length of the SPI interface short and minimize the use of vias.

Avoid routing the SPI signals over crystal oscillators (such as those generating the 27.12 MHz or 13.56 MHz frequencies), to minimize crosstalk and signal distortion.
5 Parasitic components and current return paths

During the PCB design attention must be paid to the selection of passive components and their parasitic contribution.

Most of the time component manufacturers provide measured values only at low frequency, or in a frequency range not compatible with the one used in the PCB being designed. It is then critical for a designer to know the exact value and the parasitics within the whole considered frequency range of each passive components used in the design to avoid undesired system resonance that may lead to unintentional radiating effects.

Together with the parasitic effect, the current return path on the PCB needs to be evaluated. All currents return to their source: they flow in loops, and the return path has a big impact on radiated emissions. Note that currents returning to the source follow different paths, depending upon the frequency: the path with the lowest resistance is the preferred one at low frequencies, the path with lowest inductance prevails in the high frequency range.

To understand return path resistance, it is important to calculate the impedance of a stripline and of a microstrip in the PCB, to identify paths with high and low impedance.
Decoupling and bypassing capacitors

All capacitors associated with the ST25R3916 regulator and AGD voltage pins (VSP_D, VSP_A, VSP_RF and AGD) must be as close to the chip as possible.

*Figure 12* and *Figure 13* show the typical arrangement of the decoupling capacitors, a 2.2 μF / 10 nF parallel combination.

*Figure 12. Position of the decoupling capacitors (top view)*
Figure 13. Position of the decoupling capacitors (bottom view)
Figure 14. Schematic of the decoupling capacitors
Power supply filtering

External noise may be injected into the system via power supply, and, similarly, internal noise may be conducted off of the board on the DC power leads. It is advised to put in place filters in the PCB power supply system design to reduce the effect of noise and transients on the lines. Both common and differential mode filtering are applied on the PCB.

The power supply of the ST25R3916 VDD pin is formed by a PI filter. Typical values for capacitors range from 1 to 0.01 µF, and for the ferrite bead the typical resistance value varies from 50 to 1500 Ω in the frequency range of interest (avoid saturation of ferrite bead by DC current).

![Figure 15. Detail of power supply schematic](image-url)
The thermal pad underneath the ST25R3916 provides both a ground plane and a thermal heat sink. This pad is connected to the PCB ground plane by multiple through-vias, and must be plated to have good soldering results. The multiple vias keep the total parasitic inductance low in this area.

Figure 16. ST25R3916 thermal pad
9 Mixed signal PCB layout

When a designer faces the design of a mixed signal system, the main question that comes in the PCB layout definition is related to the separation of the analog and digital parts, and consequently on how to handle the separation of ground between the two subsystems.

The origin of the split GND approach comes from the need to keep separated return currents for analog and digital subsystems. The separation between the two subsystems can be achieved by a physical separation of the two grounds (real cut) or by a spacial separation of them. When using more than one mixed signal IC with common GND connections for analog and digital signal the approach of having separate grounds may cause more issues compared to a single GND configuration.

In this case it is recommended to use only one current return plane, paying attention to partition the PCB area into digital and analog sectors; route analog and digital signals only in the analog and digital sector, respectively, to keep the return path current separated.

Additionally, the use of a split GND plane can create other problems because of the presence of possible multiple return paths of the supply current, and consequently of possible current loops. The supply current must return through a common ground terminal to which both the analog and the digital sub subsystem are referenced. The presence of the current loop will generate radiated emission, with the emission level being proportional to the loop area and to the current intensity.

A typical configuration of an 8-layer PCB implementing a full separation between analog and digital parts of the system is shown in Figure 17. In this configuration the analog components are placed on the top layer of the PCB, and the digital ones are located on the bottom layer.

Figure 18 shows a 4-layer PCB stack up configuration. Signals are routed on top and bottom layer. The inner layers are composed of a separate GND and power supply layer.
### Figure 17. Vertical separation between analog and digital signals

<table>
<thead>
<tr>
<th>Layer Name</th>
<th>Type</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top Overlay</td>
<td>Overlay</td>
<td></td>
</tr>
<tr>
<td>Top Solder</td>
<td>Solder Mask/Coverlay</td>
<td>Surface Material</td>
</tr>
<tr>
<td>Component Side - ANALOG</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric1</td>
<td>Dielectric</td>
<td>Core</td>
</tr>
<tr>
<td>Ground Plane 1 (GND) - ANALOG</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric2</td>
<td>Dielectric</td>
<td>Prepreg</td>
</tr>
<tr>
<td>Inner Layer 1 - ANALOG</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric 6</td>
<td>Dielectric</td>
<td>Core</td>
</tr>
<tr>
<td>Power Plane (VRF) - ANALOG POWER</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric5</td>
<td>Dielectric</td>
<td>Prepreg</td>
</tr>
<tr>
<td>Power Plane (VCC) - DIGITAL POWER</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric 7</td>
<td>Dielectric</td>
<td>Core</td>
</tr>
<tr>
<td>Inner Layer 3 - DIGITAL</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric 4</td>
<td>Dielectric</td>
<td>Prepreg</td>
</tr>
<tr>
<td>Ground Plane 2 (GND) - DIGITAL</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric 8</td>
<td>Dielectric</td>
<td>Core</td>
</tr>
<tr>
<td>Solder Side - DIGITAL</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Bottom Solder</td>
<td>Solder Mask/Coverlay</td>
<td>Surface Material</td>
</tr>
<tr>
<td>Bottom Overlay</td>
<td>Overlay</td>
<td></td>
</tr>
</tbody>
</table>

### Figure 18. 4-layer PCB configuration

<table>
<thead>
<tr>
<th>Layer Name</th>
<th>Type</th>
<th>Material</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top Overlay</td>
<td>Overlay</td>
<td></td>
</tr>
<tr>
<td>Top Solder</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Top Layer</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric1</td>
<td>Dielectric</td>
<td>Core</td>
</tr>
<tr>
<td>GND</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric 3</td>
<td>Dielectric</td>
<td>Prepreg</td>
</tr>
<tr>
<td>VBUS</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Dielectric 2</td>
<td>Dielectric</td>
<td>Core</td>
</tr>
<tr>
<td>Bottom Layer</td>
<td>Signal</td>
<td>Copper</td>
</tr>
<tr>
<td>Bottom Solder</td>
<td>Solder Mask/Coverlay</td>
<td>Surface Material</td>
</tr>
<tr>
<td>Bottom Overlay</td>
<td>Overlay</td>
<td></td>
</tr>
</tbody>
</table>
10 Conclusion

Many of the most common EMC problems and operational pitfalls that can be experienced when designing high power RF systems can be avoided following the guidelines illustrated in this document.

Care has to be taken especially on signal routing, and on which PCB layer the signals will be placed.

PCB stack-up, placement of components, handling of multiple DC voltages, current return path and via placement must always be an integral part of the PCB layout.

User must remember that the last step needed to achieve the required performance is often linked to an optimized PCB layout.
11 Revision history

Table 1. Document revision history

<table>
<thead>
<tr>
<th>Date</th>
<th>Revision</th>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>20-Dec-2018</td>
<td>1</td>
<td>Initial release.</td>
</tr>
<tr>
<td>26-Feb-2019</td>
<td>2</td>
<td>Document scope updated from ST Restricted to ST Public.</td>
</tr>
</tbody>
</table>
IMPORTANT NOTICE – PLEASE READ CAREFULLY

STMicroelectronics NV and its subsidiaries (“ST”) reserve the right to make changes, corrections, enhancements, modifications, and improvements to ST products and/or to this document at any time without notice. Purchasers should obtain the latest relevant information on ST products before placing orders. ST products are sold pursuant to ST’s terms and conditions of sale in place at the time of order acknowledgement.

Purchasers are solely responsible for the choice, selection, and use of ST products and ST assumes no liability for application assistance or the design of Purchasers’ products.

No license, express or implied, to any intellectual property right is granted by ST herein.

Resale of ST products with provisions different from the information set forth herein shall void any warranty granted by ST for such product.

ST and the ST logo are trademarks of ST. All other product or service names are the property of their respective owners.

Information in this document supersedes and replaces information previously supplied in any prior versions of this document.

© 2019 STMicroelectronics – All rights reserved