Introduction

The STM32WL Series microcontrollers integrate RF transceiver for LPWAN (long-power wide-area network), compatible with LoRa®, GFSK, DBPSK, and MSK, in the frequency range 150 to 960 MHz.

The STM32WL Series devices (named STM32WL later in this document) have two output powers:
- HP (high-power RFO_HP), optimized up to 22 dBm
- LP (low-power RFO_LP), optimized up to 15 dBm

The devices also include a differential RF input (RFI, up to 0 dBm) for the Rx low-noise amplifier (LNA).

To achieve the right performances for the RF output and RF input signals, some recommendations must be followed for the board design. Special care is required for the layout of an RF board compared to a conventional circuit.

This document describes precautions to be taken to achieve the best RF performance of the STM32WL on efficient applications, that last for long time under battery. The description is based on the UFBGA73 (5 x 5 mm) reference 4-layer board.
Main rules summary

Some general guidelines when routing an RF PCB are listed below:

- RF traces must be short and straightforward.
  Make the transmission lines short and straightforward in order to avoid reflections, save power and reduce high-frequency issues.
- Place and route decoupling capacitors and RF components first.
  The placement of the RF part at first, is highly recommended. Decoupling capacitors are essential to avoid high-frequency problems and maintain power integrity. Do not hesitate to add some other decoupling capacitors if needed.
- Place and route critical signals.
- Do not route high-frequency signals on board outline.
  High-frequency signals on board outline tend to radiate due to edge effects of high-frequency fields.
- Try to maintain the characteristic impedance (50 Ω) constant.
  Avoid discontinuities such as different sizes of pads put on transmission lines, bends, T-junctions, changing RF trace width along the line.
- Keep critical signals away from RF.
  High-frequency signals can induce some undesired effects in critical signals such as electric and/or magnetic coupling.
- For high-frequency applications, 4-layer PCBs are better than 2-layer.
- Try to avoid vias with RF signals.
  Vias in RF paths can cause reflections, radiation and consequently losses.
- RF return current paths must be free of obstacles or discontinuities.
- Avoid undesired magnetic coupling between inductors by leaving space between them, using magnetic shielding and/or placing them perpendicular to each other.
- Try to reduce undesired parasitic capacitances and inductances associated with the circuit layout as much as possible.
- For filter inductors such as SMPS chokes, use shielded inductors to minimizing noise and place them perpendicular to LNA traces and other RF traces.
- To reduce electromagnetic undesired emission, a metal shield can be added above RF components.

This application note applies to STM32WL Series microcontrollers based on the Arm® Cortex®-M processor.

Note: Arm is a registered trademark of Arm Limited (or its subsidiaries) in the US and/or elsewhere.
2 Characteristic and controlled impedance

All transmission lines below microwave frequencies have at least two conductors:

- In one conductor, the RF currents go towards the antenna.
- In the other, the RF currents come back to the RF source.

In order to feed an antenna, transmission lines on PCBs, designed considering their characteristic impedances, are used.

The characteristic impedance of a transmission line (sometimes represented by $Z_C$ or $Z_0$) is defined as the constant ratio between the voltage and current waves along the line. $Z_C$ can be defined with $R$, $L$, $G$ and $C$ parameters that represent the transmission line model of an extremely short segment, as shown in this formula:

$$Z_C = \frac{R + j\omega L}{G + j\omega C} = \frac{Z_{series}}{Z_{shunt}}$$

where:

- $R$ = total series resistance, per unit length of two conductors, in ohms
- $L$ = total series inductance, per unit length of two conductors, in henrys
- $G$ = shunt conductance between two conductors per unit length, in siemens
- $C$ = shunt capacitance per unit length between conductors, in farads
- $j$ = imaginary number
- $\omega$ = angular frequency, in rad/s

The impedance formed by a PCB trace and its associated reference planes, constitute the characteristic impedance of the transmission line on the PCB. This characteristic impedance on PCBs is frequently called controlled impedance.

To make it simpler, the controlled impedance of a PCB is the physical dimensions that define the $R$, $L$, $G$ and $C$ parameters. Characteristics of the materials, like permeability of permittivity, impact the value of the controlled impedance. Since no magnetic materials are used in PCBs, the relative permeability is considered equal to one ($\mu_r = 1$).

In the example of a coplanar single-ended waveguide line with lower-ground plane (GCPW for grounded coplanar waveguide), the physical dimensions like $t$ (thickness), $w$ (width), $c$ (clearance), $h$ (height) and permittivity constants of dielectric materials, determinate the characteristic impedance of the transmission line on the PCB.

Transmission lines on PCBs can also be made in other formats like microstrip or strip lines.

GCPW is often selected up to a few GHz in order to reduce radiation due to fringe fields, therefore causing less EM (electromagnetic) radiation thus less interference. For STM32WL reference boards, GCPW are used as standard transmission line structures.

GCPW is more sensitive to PCB manufacturing variations than microstrip lines. GCPW physical dimensions (such as $t$, $w$, $c$, and $h$) must be kept within low tolerances in order to maintain an impedance very close to 50 $\Omega$.
In order to understand how the manufacturing process can impact the characteristic impedance of a GCPW transmission line on a PCB, consider the example of a 4-layer PCB with physical dimensions varying with 20% tolerance, around a 50 Ω characteristic impedance at 1 GHz. In that case, the stack-up with nominal values is shown in the figure below.

**Figure 2. Example of a transmission line type GCPW on PCB**

The entire PCB stack-up for this example is depicted in the figure below.

**Figure 3. Stack-up example for 4-layer PCB**

*Note:* Due to mechanical constraints, PCBs are often made with symmetrical stack-ups.
As the transmission width varies during the manufacturing process within a 20% tolerance, the expected result is shown in the figures below.

**Figure 4. Characteristic impedance versus width variation**

**Figure 5. Characteristic impedance versus clearance variation**
The goal is to design, in theory, transmission lines that can deliver to the antenna 100% of the power inserted at the beginning of the line. To better understand the impact of the mismatch due to a characteristic impedance other than 50 Ω, see the table below.

### Table 1. Characteristic impedance and impact on RF measures (load impedance = 50 Ω)

<table>
<thead>
<tr>
<th>Characteristic impedance (Ω)</th>
<th>Reflection coefficient</th>
<th>Return loss (dB)</th>
<th>Mismatch loss (dB)</th>
<th>VSWR (1)</th>
<th>Reflected power (%)</th>
<th>Transmitted power (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>55</td>
<td>-0.048</td>
<td>0.010</td>
<td>26.444</td>
<td>1.100</td>
<td>0.23</td>
<td>99.77</td>
</tr>
<tr>
<td>54</td>
<td>-0.038</td>
<td>0.006</td>
<td>28.299</td>
<td>1.080</td>
<td>0.15</td>
<td>99.85</td>
</tr>
<tr>
<td>53</td>
<td>-0.029</td>
<td>0.004</td>
<td>30.714</td>
<td>1.060</td>
<td>0.08</td>
<td>99.92</td>
</tr>
<tr>
<td>52</td>
<td>-0.020</td>
<td>0.002</td>
<td>34.151</td>
<td>1.040</td>
<td>0.04</td>
<td>99.96</td>
</tr>
<tr>
<td>51</td>
<td>-0.010</td>
<td>0.000</td>
<td>40.086</td>
<td>1.020</td>
<td>0.01</td>
<td>99.99</td>
</tr>
<tr>
<td>50</td>
<td>0.000</td>
<td>0.000</td>
<td>-</td>
<td>1.000</td>
<td>0.00</td>
<td>100.00</td>
</tr>
<tr>
<td>49</td>
<td>0.010</td>
<td>0.000</td>
<td>39.913</td>
<td>1.020</td>
<td>0.01</td>
<td>99.99</td>
</tr>
<tr>
<td>48</td>
<td>0.020</td>
<td>0.002</td>
<td>33.804</td>
<td>1.042</td>
<td>0.04</td>
<td>99.96</td>
</tr>
<tr>
<td>47</td>
<td>0.031</td>
<td>0.004</td>
<td>30.193</td>
<td>1.064</td>
<td>0.10</td>
<td>99.90</td>
</tr>
<tr>
<td>46</td>
<td>0.042</td>
<td>0.008</td>
<td>27.604</td>
<td>1.087</td>
<td>0.17</td>
<td>99.83</td>
</tr>
<tr>
<td>45</td>
<td>0.053</td>
<td>0.012</td>
<td>25.575</td>
<td>1.111</td>
<td>0.28</td>
<td>99.72</td>
</tr>
</tbody>
</table>

1. Voltage standing wave ratio.

As a good practice, always identify the controlled impedances in schematics as depicted in the figure below.

**Figure 6. Example of schematic with controlled impedance identified**
3 RF transmission line

The geometry of a transmission line is defined to minimize the tendency of the line to act as an antenna and to radiate on its own, while the geometry of an antenna is selected to maximize its tendency to radiate.

As mentioned before, the RF transmission line on PCB is defined by its geometry and the PCB stack-up. This section includes a PCB stack-up description and some stack-ups to be copied in order to have the right impedances for the Tx and Rx paths.

3.1 Stack-up board

A typical 4-layer PCB with three types of vias is described in the figure below. The trace width, the distance between trace and ground reference, and the material characteristics determine the impedance of the RF trace. Microvias are often used with BGA packages due to the high-density interconnections (HDI).

![Figure 7. Typical 4-layer PCB stack-up with three different types of vias](image)

3.2 Stack-ups for Tx 50 \( \Omega \) and Rx 100 \( \Omega \)

One of the most difficult tasks is to correctly determine the width and clearance for an RF track from a given stack-up. The difficulty is linked to the effective dielectric constant (\( \varepsilon_{\text{eff}} \)) calculation for a given substrate.

A 2.5/3D field-solver software is often used to determine \( \varepsilon_{\text{eff}} \). PCB manufacturers can assist greatly in this task. Whenever possible, ask to the PCB manufacturer, the design rules (dimensions) to use on the RF lines to obtain 50 \( \Omega \) single-ended and 100 \( \Omega \) differential. Otherwise, copy/paste a predefined board stack-up with its characteristics and use the recommended design rules to obtain the desired impedances.

Appendix A Stack-up examples details some stack-up boards to obtain 50 \( \Omega \) for Tx lines and 100 \( \Omega \) for Rx lines that can be copied to the application. Contact the PCB manufacturer to verify if the values on the stack-up selected can be guaranteed.
3.3 Metal cutout for impedance control

For some PCB stack-up layers where prepreg are very close to external layers, it is important to reference 50 Ω lines in internal layers other than middle layer 1. In such cases, in order to maintain the 50 Ω, a metal cutout must be created in internal layers to allow RF fields to get access to the internal reference plane for which the 50 Ω impedance has been defined.

Figure 8. 50 Ω transmission lines in various layers

Figure 9. Width of metal cutout in middle layer 1 equal to the width of transmission plus clearance
Four cases exist for the cutout in internal layers. The figures below give simulation results using a known electromagnetic field solver, shown with reflection coefficients (S11) in the Smith chart, considering four w cases for the cutout in internal layers (shown in Figure 9).

Figure 10. Example 1: internal cutout equal to w (width)

Figure 11. Example 2: internal cutout greater than w
From these simulation results, the best cutout is the first one, with the cutout width equal to the RF line, plus clearance on top layer, as shown in Figure 9.
4 Surface mounted components with RF signals

4.1 Capacitors

The table below gives some recommendations regarding the routing of SMD (surface mounted components).

<table>
<thead>
<tr>
<th>Performance</th>
<th>Capacitor pad type</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td><img src="image1" alt="Recommended" /></td>
<td>Short traces with multiples vias reducing return current impedance</td>
</tr>
<tr>
<td>Better</td>
<td><img src="image2" alt="Better" /></td>
<td>Short traces</td>
</tr>
<tr>
<td>Poor</td>
<td><img src="image3" alt="Poor" /></td>
<td>Long traces between capacitor increasing series inductance</td>
</tr>
<tr>
<td>Not good</td>
<td><img src="image4" alt="Not good" /></td>
<td>Thinner access track increasing the equivalent series inductance of the capacitor</td>
</tr>
</tbody>
</table>
Whenever possible, thermal reliefs must be avoided on RF lines as they increase the equivalent series inductance (ESL) of capacitors and then change the frequency response of the capacitors in addition to increasing losses.
4.2 Inductors

The table below gives some recommendations regarding the inductors.

**Table 3. Inductor pads with RF signals**

<table>
<thead>
<tr>
<th>Performance</th>
<th>Inductor pad type</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td><img src="image" alt="Recommended" /></td>
<td>Short and same PAD width access traces, maintaining the original value of the inductance and Q-Factor</td>
</tr>
<tr>
<td>Not good</td>
<td><img src="image" alt="Not good" /></td>
<td>Be careful with this kind of tricks. This narrow trace contributes to increase the inductance, but this can decrease the equivalent Q-factor of the inductor. RF inductors are carefully made to have a high Q-factor. Do not ruin it.</td>
</tr>
</tbody>
</table>
5 Via stitching and shielding

The recommendation is to put some vias around RF lines as shown in the figure below, in order to reduce high-frequency issues.

Figure 16. Spacing between vias around GCPW

The following formula is used to determine the D value:

\[
\frac{\lambda_G}{20} \leq D \leq \frac{\lambda_G}{10}
\]

with \( \lambda_G \), as guided wavelength, defined by this formula:

\[
\lambda_G = \frac{3 \cdot 10^8}{f \cdot \sqrt{\varepsilon_{r_{eff}}}}
\]

where:
- \( f \) = highest frequency of the RF circuit operation
- \( \varepsilon_{r_{eff}} \) = effective dielectric constant of the PCB
6 RF return current path

The RF currents that go to the antenna must come back to their source inside the chip to complete a closed loop: it is done by a return path. Thus, a return path for the delivery medium back to the energy source must be provided. A return path is defined as the conductive path taken by the current returning to the source from the load, generally this return path is done on a grounded plane.

Table 4. Return paths

<table>
<thead>
<tr>
<th>Performance</th>
<th>Return path type</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td><img src="image1" alt="Diagram" /></td>
<td>No vias in RF return path</td>
</tr>
<tr>
<td></td>
<td>Make clean current return paths beneath RF Lines</td>
<td></td>
</tr>
<tr>
<td>Not good</td>
<td><img src="image2" alt="Diagram" /></td>
<td>Vias creating larger RF return path increasing losses and discontinuities</td>
</tr>
<tr>
<td></td>
<td>It is a puzzle for currents that search their return paths</td>
<td></td>
</tr>
</tbody>
</table>
Figure 17. Clean return path example for RF currents

Top Layer

Middle Layer 2
Discontinuities to avoid in transmission lines

When designing transmission line on PCB with a controlled impedance (50 Ω), the objective is to maintain the same impedance in the whole system in order to transfer as much as energy as possible to the antenna and to minimize the unintentional loss of energy in the transmission line.

<table>
<thead>
<tr>
<th>Performance</th>
<th>Layout</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Poor</td>
<td><img src="image1" alt="Image" /></td>
<td>Difference between component pad widths and RF line widths, thermal reliefs and components placed in a way creating parasitic effects</td>
</tr>
</tbody>
</table>

Taking the above routing and increase slightly layout dimensions allow the user to route the RF lines without discontinuities as shown below.

| Recommended | ![Image](image2) | Clean RF lines with pad components at the same width as the RF lines and pad components on the RF lines |

Table 5. Layout discontinuities
### Table 6. Track transitions

<table>
<thead>
<tr>
<th>Performance</th>
<th>Transition type</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td>Smooth transition</td>
<td></td>
</tr>
<tr>
<td>Poor</td>
<td>Multi-step transition</td>
<td></td>
</tr>
<tr>
<td>Not good</td>
<td>Single-step transition</td>
<td></td>
</tr>
</tbody>
</table>

### Table 7. Test points

<table>
<thead>
<tr>
<th>Performance</th>
<th>Test point type</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td>Test point inside the RF line (avoiding stubs)</td>
<td>Test point with <strong>no</strong> stub</td>
</tr>
<tr>
<td>Not good</td>
<td>Test point acting as a stub</td>
<td>Test point as <strong>stub</strong></td>
</tr>
</tbody>
</table>
Whenever possible, align the width between the RF lines and pads (no transition needed). Do not hesitate to reduce pad components to maintain constant width of RF traces.

**Table 8. Pad component width**

<table>
<thead>
<tr>
<th>Performance</th>
<th>Pad component type</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td>Same width for RF line and pads</td>
<td>No transition needed</td>
</tr>
<tr>
<td>Better</td>
<td>Tapered transition</td>
<td>Smooth transition</td>
</tr>
<tr>
<td>Not good</td>
<td>Abrupt transition with different width</td>
<td>Single-step transition</td>
</tr>
</tbody>
</table>

**Table 9. RF switch transitions**

<table>
<thead>
<tr>
<th>Performance</th>
<th>RF switch transition type</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td>Smooth transition</td>
<td></td>
</tr>
<tr>
<td>Better</td>
<td>Tapered transition</td>
<td></td>
</tr>
<tr>
<td>Not good</td>
<td>Single-step transition</td>
<td></td>
</tr>
<tr>
<td>Performance</td>
<td>Pad to RF line transition type</td>
<td>Comment</td>
</tr>
<tr>
<td>-------------</td>
<td>-------------------------------</td>
<td>---------</td>
</tr>
<tr>
<td><strong>Recommended</strong></td>
<td><img src="image" alt="Smooth transition with polygons" /></td>
<td>Smooth transition with polygons</td>
</tr>
<tr>
<td><strong>Not good</strong></td>
<td><img src="image" alt="Thin traces causing losses in high frequencies" /></td>
<td>Thin trace with single-step transition</td>
</tr>
<tr>
<td></td>
<td><img src="image" alt="Single-step transition" /></td>
<td>Single-step transition</td>
</tr>
</tbody>
</table>
8  Bends with RF lines

A bend is needed when there is a direction change for an RF line. Bends with RF lines can cause reflections and power loss. Some guidelines are detailed in this section to avoid issues with bends in high-frequency transmission lines. The main idea when designing bends is to keep the same trace width in the corner.

Consider the worst case that is the 90° bend shown in the figure below.

Figure 18. 90° bend example
![Figure 18](image)

The ideal case is a straight line with a constant width as shown below.

Figure 19. Ideal case: straight line
![Figure 19](image)
Table 11. Guidelines for bends in RF lines

<table>
<thead>
<tr>
<th>Performance</th>
<th>Bend type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td><img src="image" alt="Recommended Bend" /></td>
</tr>
<tr>
<td>Better</td>
<td><img src="image" alt="Better Bend" /></td>
</tr>
<tr>
<td>Better</td>
<td><img src="image" alt="Better Bend" /></td>
</tr>
<tr>
<td>Not good</td>
<td><img src="image" alt="Not Good Bend" /></td>
</tr>
</tbody>
</table>
9 Minimize unintentional radiation

9.1 RFO harmonics

The typical RFO application circuit for the STM32WL is shown in the figure below.

![Figure 20. Typical circuit for RFO harmonics](image)

The STM32WL features a linear, high-efficiency RF PA (power amplifier) connected to the RFO pin (PA output). Due to the high-frequency harmonic components generated at RFO (above GHz for an operating frequency starting at 500 MHz), the RF tracks before filtering stages (before L5, C21, C24, C22, L6 and C25 in the schematic) may radiate unintentional electromagnetic (EM) energy. Any piece of metal that makes λ/4 under certain conditions, can act as an antenna radiating EM energy.

**Note:** Remember that the power radiated by a linear antenna of length L, is proportional to \( P = (L/\lambda)^2 \). This means that the bigger the unintentional antenna is, the greater the amount of energy it radiates.

The following formula can be used to determine the longest length of a track to not radiate EM energy on a PCB:

\[
L < \frac{3 \cdot 10^8}{4 \cdot h \cdot f \cdot \sqrt{\varepsilon_{r,\text{eff}}}}
\]

where:
- \( h \) is the harmonic for which the user must determine the maximum track length to avoid.
- \( f \) is the operating frequency of the RF signal.
- \( \varepsilon_{r,\text{eff}} \) is the effective dielectric constant of the PCB stack-up layers.

**Example**

For an operating frequency at 915 MHz, the ninth harmonic (h9) is equal to 8.235 GHz (9 x 915 MHz). For PCB with an \( \varepsilon_{r,\text{eff}} = 3 \), the maximum track length is:

\[
L < \frac{3 \cdot 10^8}{4 \cdot 9 \cdot 915 \cdot 10^6 \cdot \sqrt{3}} = 5.258 \text{ mm}
\]

The maximum track length to avoid an unintentional harmonic radiation with an operating frequency at 915 MHz and taking the ninth harmonic, is 5.258 mm.
9.2 High-frequency signals on board outline

Routing high-frequency signals at board outline may cause unintentional EM radiation.

Figure 21. EM radiation generated by HF signals

[Diagram showing EM radiation generated by HF signals]

One solution to mitigate the problem of tracks that radiate EM is to place them between grounded planes (below and above).

Figure 22. How to mitigate unintentional EM radiation

[Diagram showing mitigation of unintentional EM radiation]

Vias to ground added with a guard trace on top, in order to ‘absorb’ the fringe fields.
9.3 Ground flooding
Flooding unused PCB areas with GND and with multiple vias, can be used to keep the GND impedance low and reduce EMC issues.

Figure 23. PCB example with or without ground flooding

9.4 Metal shield
To prevent issues due to unintentional radiation of harmonic contents, it is highly recommended to put a metal shield to cover the RF part on the board.

Figure 24. PCB example with or without metal shield

High harmonics may cause EMC issues.
The metal shield prevents harmonic components from causing interference with other circuits.
9.5 Power planes

To prevent unintentional EM radiation between GND planes and power planes, the power planes must not be routed at the edge of the board. Otherwise these power planes may radiate unintentional EM due to fringe fields. GND planes must be put in all layers around the board and must be connected together.

**Figure 25. GND and power planes**

- GND guard trace in the same layer than the power plane

In this example:
- power plane in Layer 2 ending before the edge
- GND plane in Layer 3 with guard trace in Layer 2
Decoupling capacitors

Capacitors with lower values must be placed closer to the chip than higher-value ones, as shown in the figure below.

![Figure 26. Decoupling capacitors](image)

When routing decoupling capacitors, the smallest possible current loop must be maintained. Large current loops are translated into inductive behavior.

<table>
<thead>
<tr>
<th>Performance</th>
<th>RF switch transition type</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Recommended</td>
<td><img src="image" alt="Current loop" /></td>
<td>Reduced current loop</td>
</tr>
<tr>
<td>Poor</td>
<td><img src="image" alt="Current loop" /></td>
<td>Large current loop</td>
</tr>
</tbody>
</table>

The equivalent series inductance (ESL, see the figure below) of a capacitor is impacted by the current loop.

![Figure 27. High-frequency equivalent model of a capacitor](image)
The reference PCB 4-layer layout for BGA package is detailed in the figures below.

**Figure 28. All layers of STM32WL reference layout for BGA**

**Figure 29. Top layer of STM32WL reference layout for BGA**
Figure 30. Middle layer 1 of STM32WL reference layout for BGA

Figure 31. Middle layer 2 of STM32WL reference layout for BGA
Figure 32. Bottom layer of STM32WL reference layout for BGA
Documentation references

• Advanced Design System 2020, Keysight Technologies.
13 Conclusion

Some care must be taken when designing an RF board. Guidelines for decoupling capacitors, RF general rules, reduction of EMC issues, controlled impedances with predefined PCB stack-up layers are presented in this application note. The user must adapt these guidelines to the application.

Those guidelines must be followed to secure a correct behavior of the application, with high performance for the RF part of the STM32WL board.
Appendix A  Stack-up examples

Some stack-up examples to obtain 50 Ω for Tx lines and 100 Ω for Rx lines from a typical stack-up for BGA package as shown in the figure below.

**Figure 33. Typical stack-up for BGA package**

- **Case 1**: typical stack-up for BGA package with **PCB total thickness = 1.04 mm**
  Consider configuration detailed in the table below.

**Table 13. Case 1: PCB total thickness = 1.04 mm**

<table>
<thead>
<tr>
<th>Element</th>
<th>Material</th>
<th>Nominal thickness h_x (μm)</th>
<th>ε_r</th>
<th>Layer</th>
<th>Nominal thickness t (μm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solder mask (h_3)</td>
<td>Solder resist</td>
<td>20</td>
<td>3.7</td>
<td>Top</td>
<td>35</td>
</tr>
<tr>
<td>Prepreg 1 (h_1)</td>
<td>1 x 2116</td>
<td>70</td>
<td>3.5</td>
<td>Middle 1 and middle 2</td>
<td>35</td>
</tr>
<tr>
<td>Core (h_2)</td>
<td>FR4</td>
<td>710</td>
<td>5.0</td>
<td>Bottom</td>
<td>35</td>
</tr>
</tbody>
</table>

The Tx and Rx lines detailed in the figures below can then be built from this configuration.

**Figure 34. Tx 50 ohms RF tracks (case 1, PCB total = 1.04 mm)**
Figure 35. Rx 100 ohms differential pair (case 1, PCB total = 1.04 mm)

- **Case 2**: typical stack-up for BGA package with PCB total thickness = 1.10 mm
  Consider configuration detailed in the table below.

**Table 14. Case 2: PCB total thickness = 1.10 mm**

<table>
<thead>
<tr>
<th>Dielectric materials</th>
<th>Metal layers</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Element</td>
<td>Material</td>
<td>Nominal thickness h_x (μm)</td>
<td>ε_r</td>
</tr>
<tr>
<td>Solder mask (h_3)</td>
<td>solder resist</td>
<td>20</td>
<td>3.3</td>
</tr>
<tr>
<td>Prepreg 1 (h_1)</td>
<td>1 x 2116</td>
<td>108</td>
<td>3.8</td>
</tr>
<tr>
<td>Core (h_2)</td>
<td>FR4</td>
<td>710</td>
<td>5.0</td>
</tr>
</tbody>
</table>

The Tx and Rx lines detailed in the figures below can then be built from this configuration.

Figure 36. Tx 50 Ω RF tracks (case 2, PCB total = 1.10 mm)
• **Case 3**: typical stack-up for BGA package with **PCB total thickness = 1.60 mm**
  Consider configuration detailed in the table below.

**Table 15. Case 3: PCB total thickness = 1.60 mm**

<table>
<thead>
<tr>
<th>Element</th>
<th>Material</th>
<th>Nominal thickness $h_x$ ($\mu$m)</th>
<th>$\varepsilon_r$</th>
<th>Layer</th>
<th>Nominal thickness $t$ ($\mu$m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solder mask ($h_3$)</td>
<td>solder resist</td>
<td>20</td>
<td>3.5</td>
<td>Top</td>
<td>35</td>
</tr>
<tr>
<td>Prepreg 1 ($h_1$)</td>
<td>1 x 1080</td>
<td>76</td>
<td>4.18</td>
<td>Middle 1and 2</td>
<td>35</td>
</tr>
<tr>
<td>Core ($h_2$)</td>
<td>7 x 7628</td>
<td>1268</td>
<td>4.74</td>
<td>Bottom</td>
<td>35</td>
</tr>
</tbody>
</table>

The Tx and Rx lines details in the figures below can then be built from this configuration.
Important:
The longer the distance is between the source and the antenna, the greater the potential for loss of energy in the RF transmission line. As a design rule, RF transmission lines must be as short as possible and without discontinuities.
Revision history

Table 16. Document revision history

<table>
<thead>
<tr>
<th>Date</th>
<th>Version</th>
<th>Changes</th>
</tr>
</thead>
<tbody>
<tr>
<td>6-Mar-2020</td>
<td>1</td>
<td>Initial release.</td>
</tr>
</tbody>
</table>
List of tables

Table 1. Characteristic impedance and impact on RF measures (load impedance = 50 Ω) ........................................... 6
Table 2. Capacitor pads with RF signals .................................................................................................................... 11
Table 3. Inductor pads with RF signals ..................................................................................................................... 13
Table 4. Return paths ............................................................................................................................................... 15
Table 5. Layout discontinuities ............................................................................................................................... 17
Table 6. Track transitions ......................................................................................................................................... 18
Table 7. Test points .................................................................................................................................................. 18
Table 8. Pad component width ................................................................................................................................ 19
Table 9. RF switch transitions ................................................................................................................................... 19
Table 10. Package pad to RF line transitions ........................................................................................................ 20
Table 11. Guidelines for bends in RF lines ................................................................................................................ 22
Table 12. Return currents for decoupling capacitors ............................................................................................. 27
Table 13. Case 1: PCB total thickness = 1.04 mm .................................................................................................... 34
Table 14. Case 2: PCB total thickness = 1.10 mm ..................................................................................................... 35
Table 15. Case 3: PCB total thickness = 1.60 mm ..................................................................................................... 36
Table 16. Document revision history ....................................................................................................................... 38
List of figures

Figure 1. Example of a GCPW in a 2-layer PCB .......................................................... 3
Figure 2. Example of a transmission line type GCPW on PCB ................................. 4
Figure 3. Stack-up example for 4-layer PCB ............................................................ 4
Figure 4. Characteristic impedance versus width variation ...................................... 5
Figure 5. Characteristic impedance versus clearance variation ............................... 5
Figure 6. Example of schematic with controlled impedance identified .................... 6
Figure 7. Typical 4-layer PCB stack-up with three different types of vias .................... 7
Figure 8. 50 Ω transmission lines in various layers ................................................... 8
Figure 9. Width of metal cutout in middle layer 1 equal to the width of transmission plus clearance 8
Figure 10. Example 1: internal cutout equal to w (width) ......................................... 9
Figure 11. Example 2: internal cutout greater than w ............................................. 9
Figure 12. Example 3: internal cutout smaller than w ........................................... 10
Figure 13. Example 4: internal cutout much smaller than w .................................... 10
Figure 14. Example of capacitors on RF lines .......................................................... 12
Figure 15. Thermal reliefs ...................................................................................... 12
Figure 16. Spacing between vias around GCPW ...................................................... 14
Figure 17. Clean return path example for RF currents .............................................. 16
Figure 18. 90° bend example .................................................................................. 21
Figure 19. Ideal case: straight line ........................................................................... 21
Figure 20. Typical circuit for RFO harmonics ......................................................... 23
Figure 21. EM radiation generated by HF signals ..................................................... 24
Figure 22. How to mitigate unintentional EM radiation ........................................... 24
Figure 23. PCB example with or without ground flooding ........................................ 25
Figure 24. PCB example with or without metal shield .............................................. 25
Figure 25. GND and power planes .......................................................................... 26
Figure 26. Decoupling capacitors ........................................................................... 27
Figure 27. High-frequency equivalent model of a capacitor ...................................... 27
Figure 28. All layers of STM32WL reference layout for BGA ................................. 28
Figure 29. Top layer of STM32WL reference layout for BGA ................................. 28
Figure 30. Middle layer 1 of STM32WL reference layout for BGA ......................... 29
Figure 31. Middle layer 2 of STM32WL reference layout for BGA ......................... 29
Figure 32. Bottom layer of STM32WL reference layout for BGA ......................... 30
Figure 33. Typical stack-up for BGA package .......................................................... 34
Figure 34. Tx 50 ohms RF tracks (case 1, PCB total = 1.04 mm) ............................... 34
Figure 35. Rx 100 ohms differential pair (case 1, PCB total = 1.04 mm) .................... 35
Figure 36. Tx 50 Ω RF tracks (case 2, PCB total = 1.10 mm) ..................................... 35
Figure 37. Rx 100 Ω differential pair (case 2, PCB total = 1.10 mm) ....................... 36
Figure 38. Tx 50 Ω RF tracks (case 3, PCB total = 1.60 mm) ..................................... 36
Figure 39. Rx 100 Ω differential pair (case 3, PCB total = 1.60 mm) ....................... 37