CC26xx HW Training

Layout Considerations

Fredrik Kervel, Bluetooth Smart Applications
Reference Schematic

VDDS Decoupling Capacitors
- Pin 11
- Pin 28
- Pin 18

VDDR Decoupling Capacitors
- Pin 29
- Pin 32

Place L1 and C8 close to pin 17.

R12 and C15 for antenna matching.
Follow the reference layout! 😊

- All reference designs are for 2-layer PCBs
  - Thickness = 0.8 mm
  - 4 (or more) layers is also ok

- Place the RF match close to the RF pins

- Solid ground plane
  - No signal traces underneath the RF path!
  - Ground return paths between the antenna / RF-components and CC26xx must be uninterrupted
  - Keep as much signal- and power routing on the top layer as possible

- Place decoupling caps as close to the VDD pins as possible
  - Ground return paths between decoupling caps and CC26xx should be short and direct

- DC/DC-regulator must have a short and direct ground connection to CC26xx
Reference Layout – Differential output

- Antenna match components
- Longer RF traces must have 50 ohm impedance
- Notice orientation of pi-filter layout
  - Shunt components oriented opposite way to avoid crosstalk
Reference Layout – Differential output

- Antenna match components
- Longer RF traces must have 50 ohm impedance
- Notice orientation of pi-filter layout
  - Shunt components oriented opposite way to avoid crosstalk

- No traces underneath the RF path
  - Will increase the impedance of the RF ground return paths and, even worse, create current loops
  - May lead to reduced RF performance and spurious emission
Reference Layout – Single ended output
Reference Layout – Everything else

- Make sure decoupling ground paths are short and direct (low impedance)
- Make sure the DCDC switch ground path is short and direct (low impedance)
- Try to locate as much routing as possible on the top layer in 2-layer PCBs
• Make sure decoupling ground paths are short and direct (low impedance)

• Make sure the DCDC switch ground path is short and direct (low impedance)

• Try to locate as much routing as possible on the top layer in 2-layer PCBs
For RF traces longer than a certain length the impedance should be controlled.

TXLine is a free tool for PCB trace impedance calculations