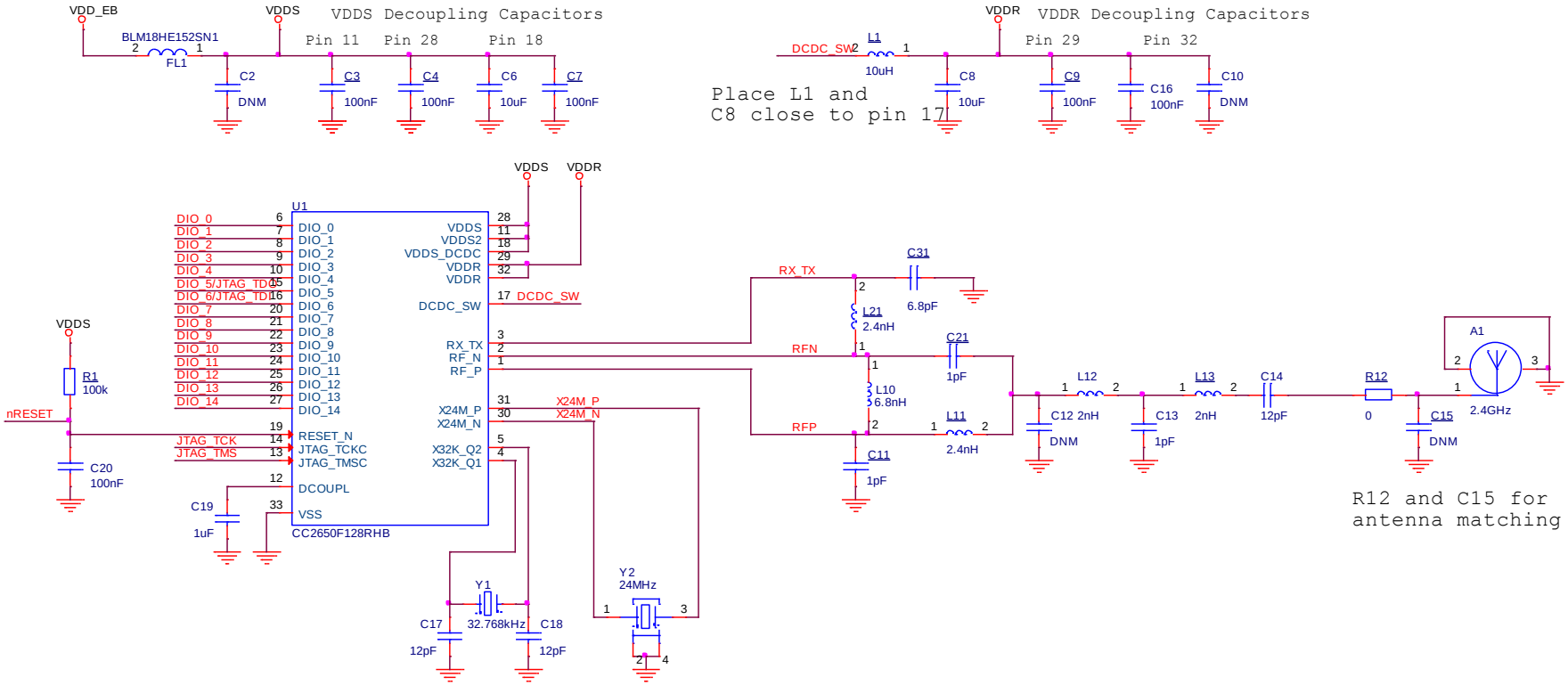
CC26xx HW Training

Layout Considerations

Fredrik Kervel, Bluetooth Smart Applications

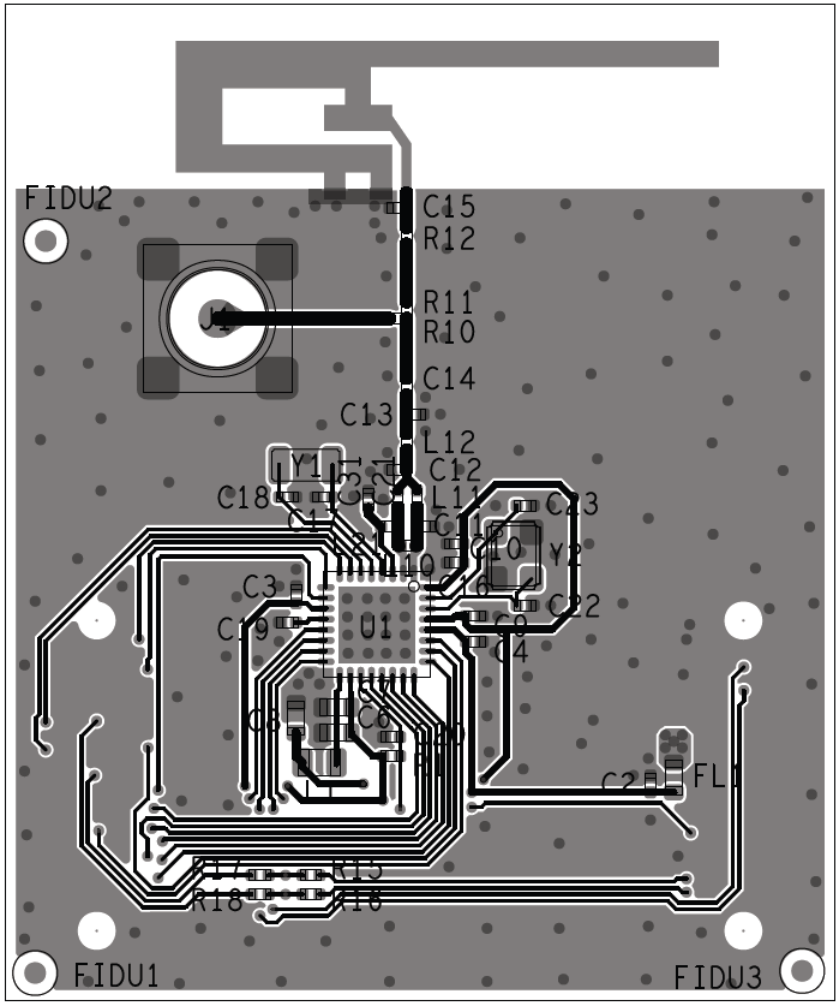
Texas Instruments

Reference Schematic



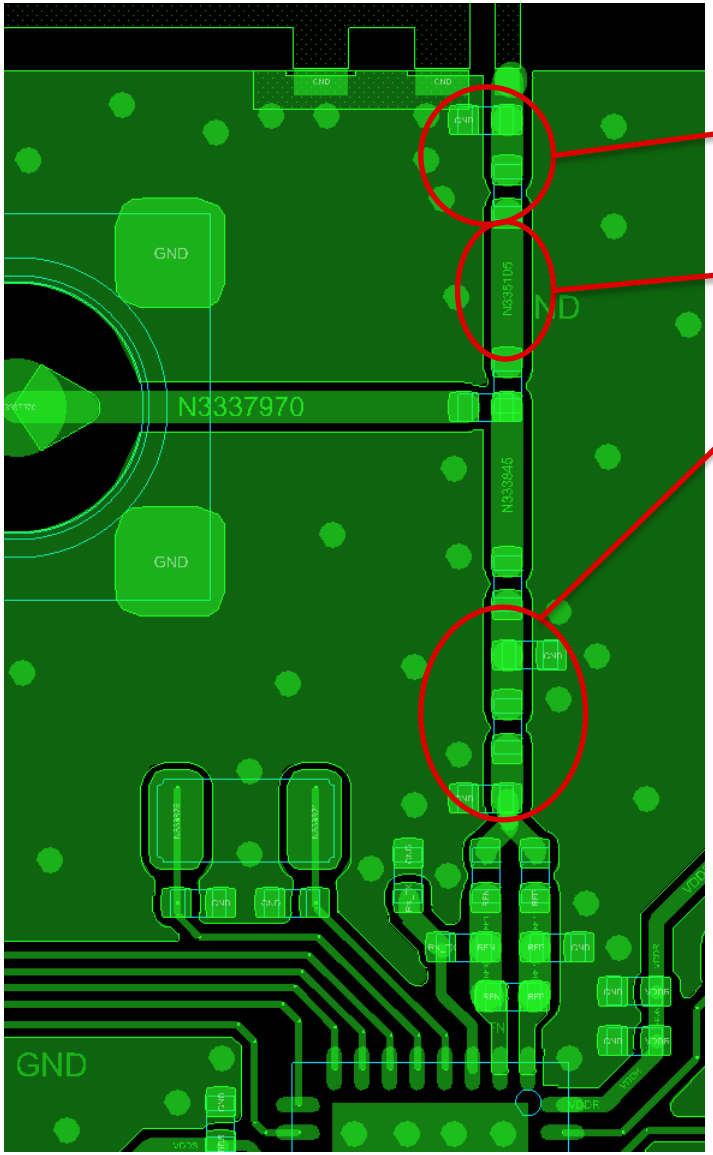
Texas Instruments

Reference Layout



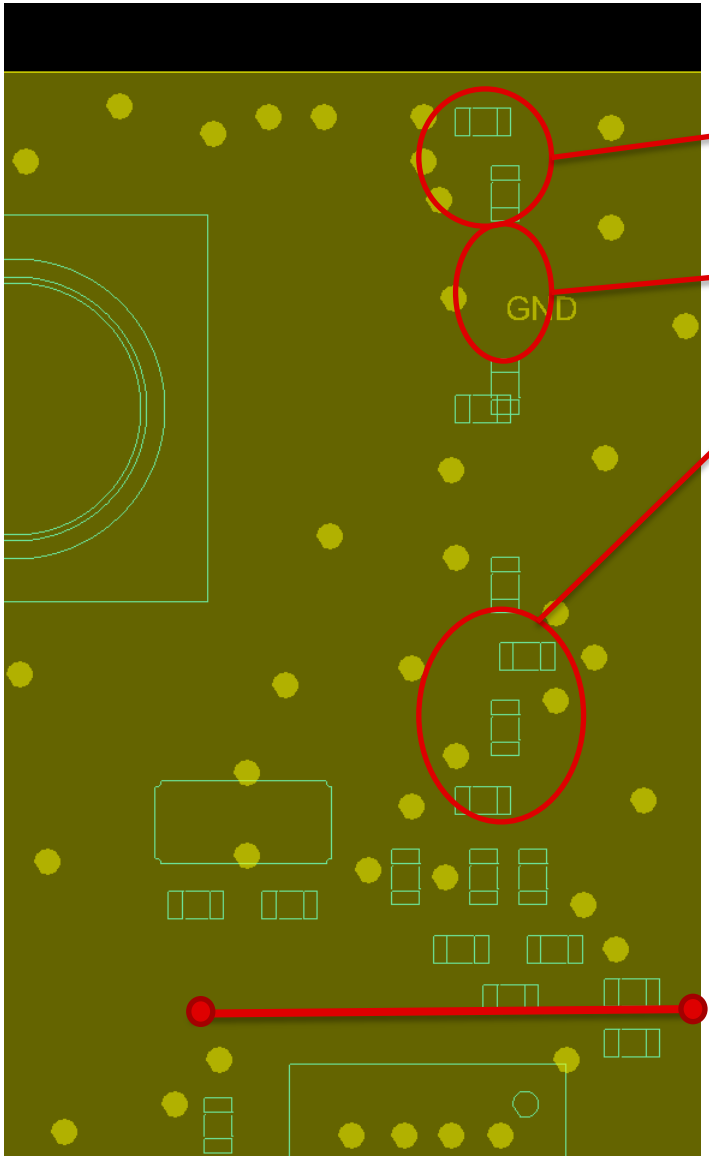
- Follow the reference layout! ☺
- All reference designs are for 2-layer PCBs
  - Thickness = 0.8 mm
  - 4 (or more) layers is also ok
- Place the RF match close to the RF pins
- Solid ground plane
  - No signal traces underneath the RF path!
  - Ground return paths between the antenna / RF-components 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



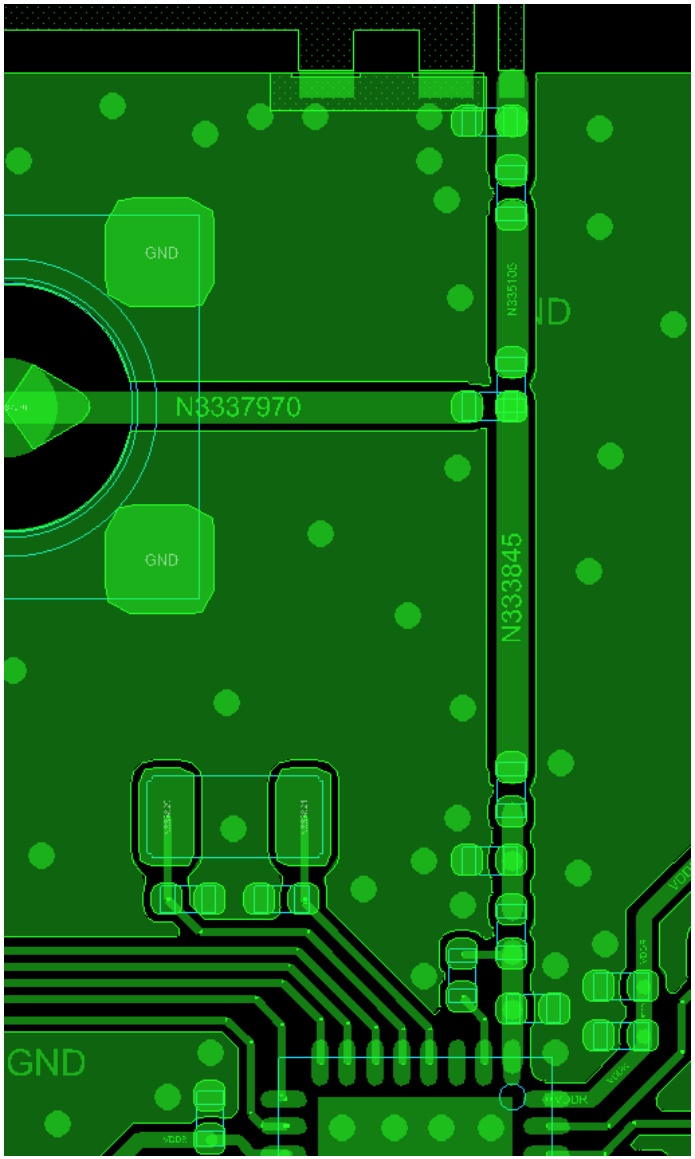
- Antenna match components
- Longer RF traces must have 50 ohm impedance
- Notice orientation of pi-filter layout
  - Shunt components oriented opposite way to avoid crosstalk

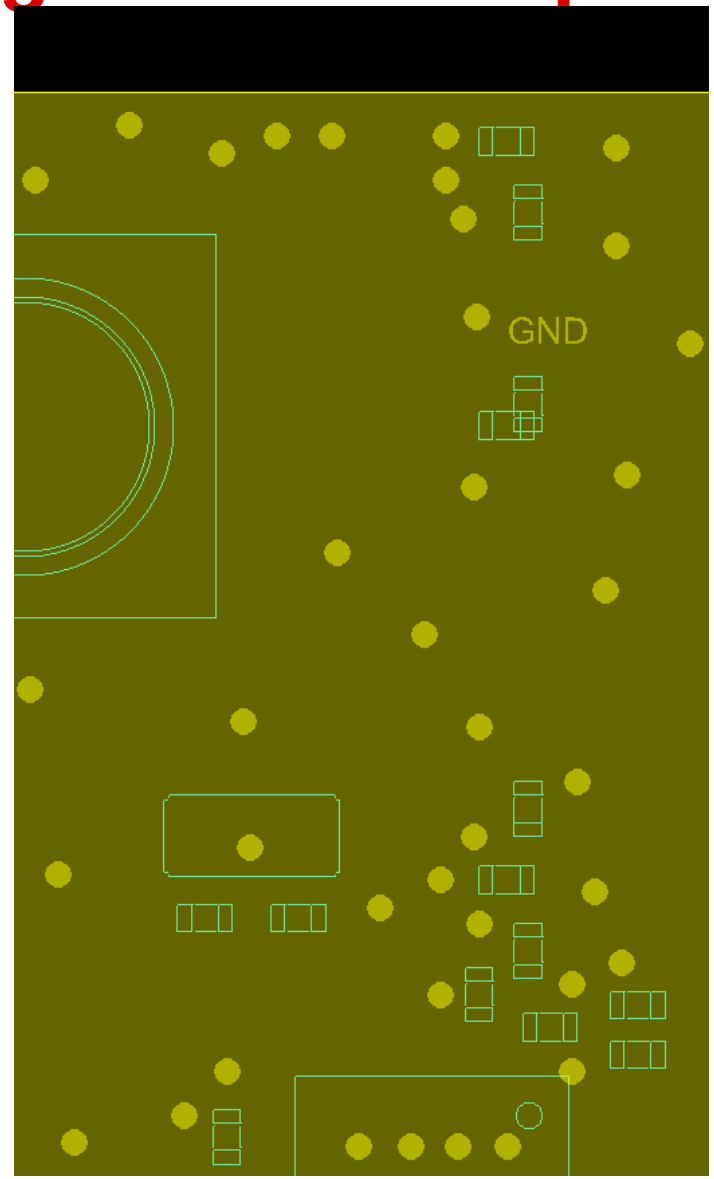
Reference Layout – Differential output



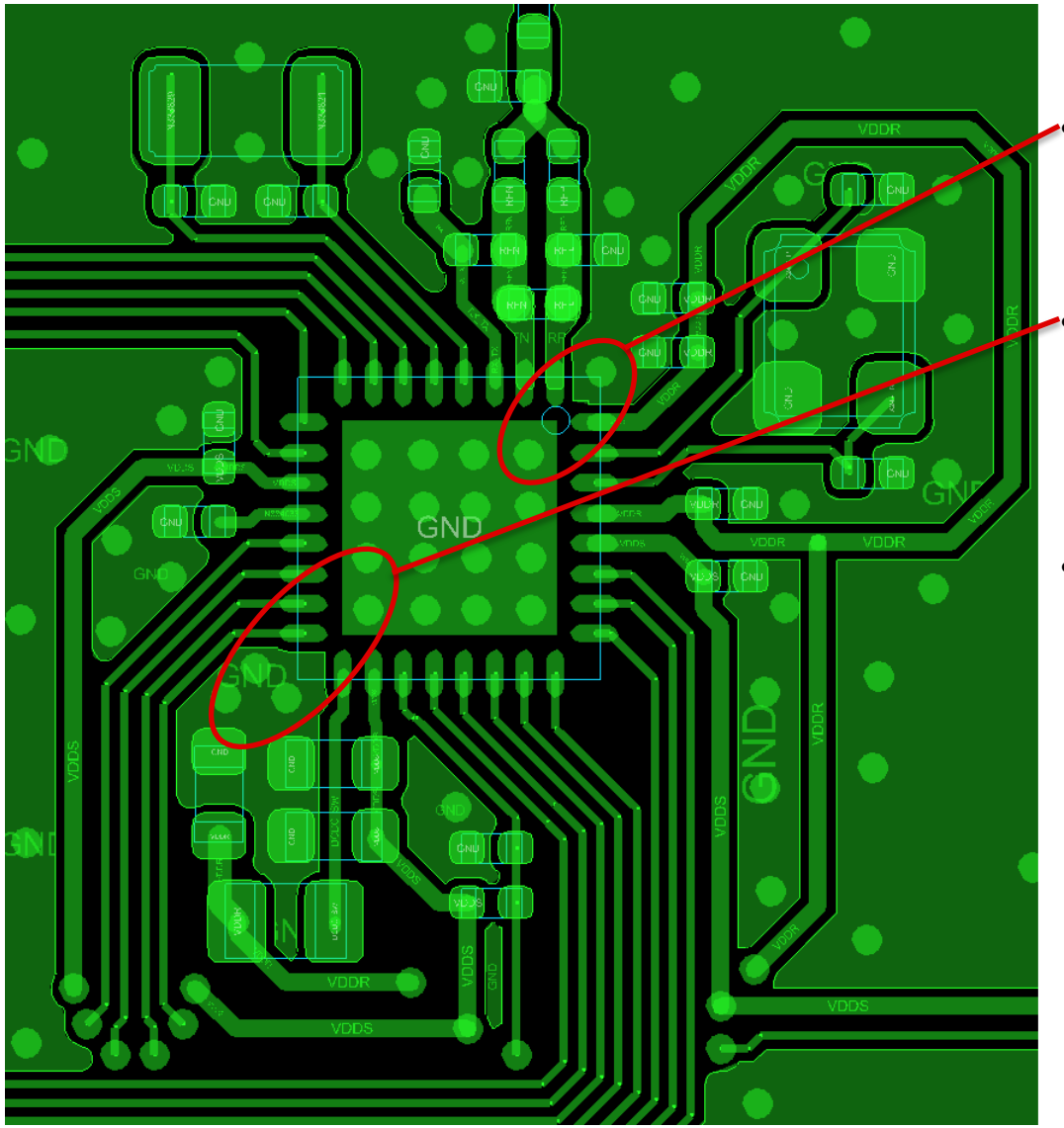
- Antenna match components
- Longer RF traces must have 50 ohm impedance
- Notice orientation of pi-filter layout
  - Shunt components oriented opposite way to avoid crosstalk
- No traces underneath the RF path
  - Will increase the impedance of the RF ground return paths and, even worse, create current loops
  - May lead to reduced RF performance and spuriuos emission

Reference Layout – Single ended output



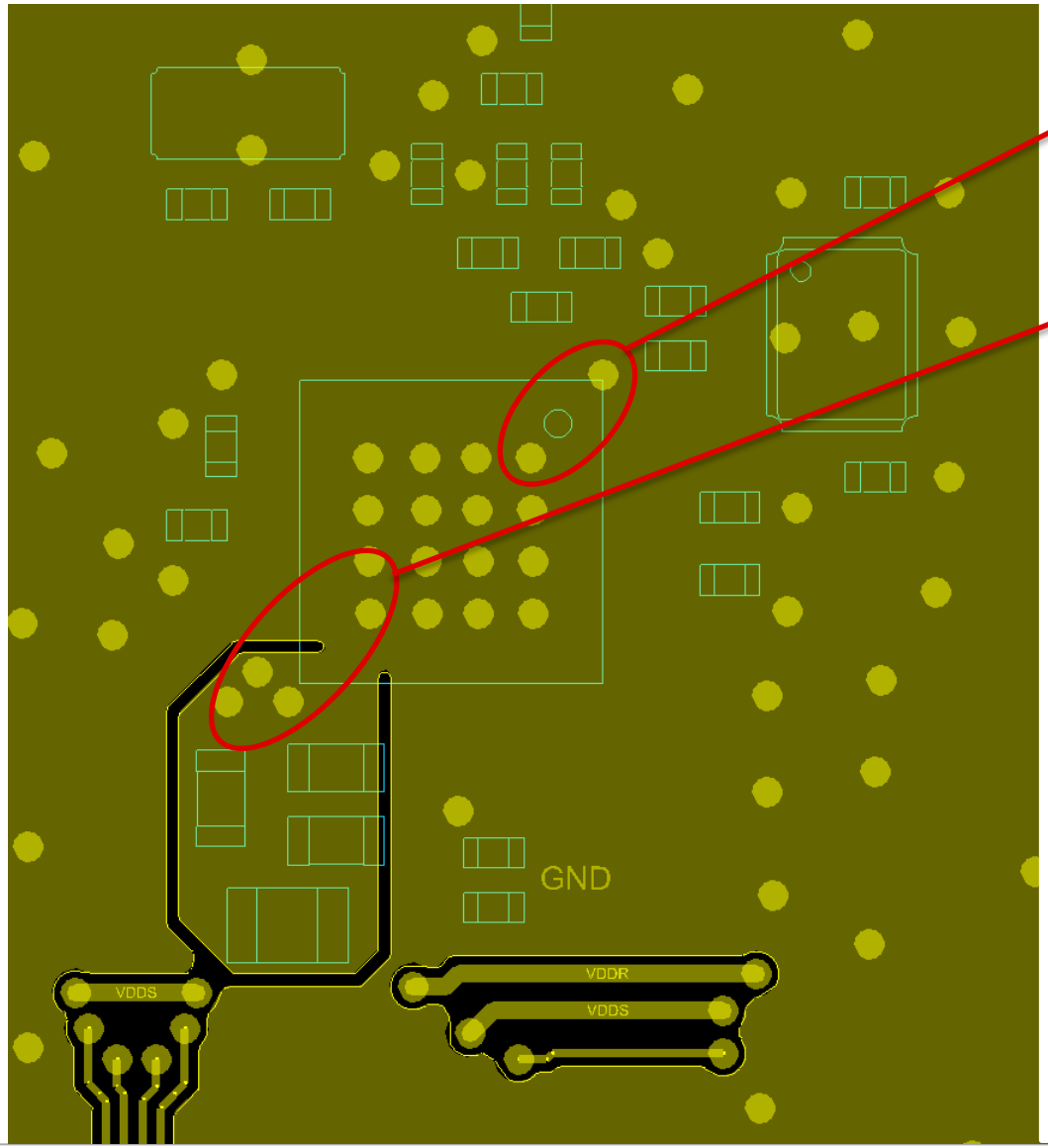


Reference Layout – Everything else



- 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

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

Reference Layout – Trace impedance

- For RF traces longer than a certain length the impedance should be controlled

- TXLine is a free tool for PCB trace impedance calculations
  - http://www.awrcorp.com/products/optional-products/tx-line-transmission-line-calculator