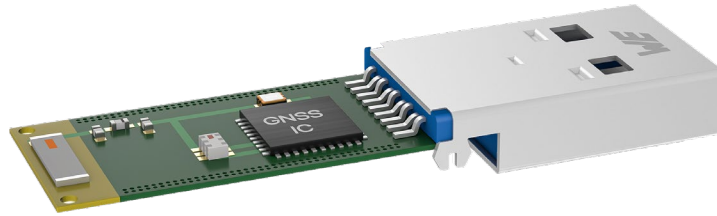# Antenna Design-In Recommendations



These recommendations form the basis for implementing our chip antennas according to the specified antenna performance in the application:

- Consider the recommended PCB layout for the antenna from the antenna data sheet
- Cut out the copper in all PCB layers under the antenna
- Place electrical antennas separated from the rest of the circuit and components ([ANP057](#))
- Place magnetic antennas as centrally as possible at the edge of the longer PCB side
- Do not place any electroconductive planes, metals and batteries/accumulators in the immediate environment of the antenna ([ANP057](#))
- Maximize the distance of the antenna to dielectric materials (e.g. casing)
- Consider the recommended PCB layout for the matching circuit from the antenna data sheet
- Place the matching circuit as close as possible to the antenna and connect it to the radio module ground plane
- Design the matching circuit for component size 0402
- Design the feed line of the antenna to the impedance of 50 Ohm ([ANE012](#))
- Keep the feed line of the antenna as short as possible
- Provide the footprint for a [WR-UMRF connector](#) on the feed line of the antenna immediately after the circuit of the radio module
- Consider the recommended circuit and the PCB layout of the antenna feed line from the radio module data sheet

For more information, please refer to our Application Notes and Design Guides:

- [ANP057: WE-MCA Multilayer Chip Antenna Placement & Matching](#)
- [ANR017 GNSS Antenna Selection](#)
- [ANP074: Introduction to RF inductors](#)
- [ANE012: Coaxial PCB Connector - PCB-Transmission Line Design Guide](#)
- [ANR022: Radio module cross migration guide](#)
- [ANR010: Range Estimation](#)