

CHIP ANTENNA INTEGRATION

Fractus Antennas experts can review any customer design (layout, schematics, etc.) and provide recommendations to maximize the antenna performance. Fractus Antennas expertise and resources can reduce development costs and improve time to market. We ensure the best antenna solution is recommended for every specific device.

In order to integrate a Fractus Antennas product, the following practices are recommended:

Clearance area and volume: Keep an area free from electronic components, traces and ground plane in all PCB layers and the underside of the PCB directly underneath the mounted antenna. Maximize the volumetric clearance -the distance between the antenna and the closest mechanic part-.

Antenna Location: Place the antenna on the corner of the PCB, as far as possible from other components, such as LCDs, batteries, connectors, especially those components and covers with metallic characteristics.

Matching Network: Place pads for the matching network compatible with 0402/0603 SMD components, as close as possible to the antenna feed point. Do it in the ground plane area, not in the clearance area. This provides a degree of freedom to tune the antenna once the design is finished and all the elements of the system (batteries, displays, covers, etc.) are in place.

Transmission line and RF Chip: Design your transmission line so that its characteristic impedance is equal to 50Ω , as well as the output impedance of the RF chip. Locate your RF chip as close as possible to the matching network in order to reduce the losses introduced by the transmission line.

Area underneath the Matching Network: if possible avoid any ground plane area underneath the pads of the matching network in any layer placed at a distance shorter than 1 mm of the PCB.