



PAGE 2/8

ISSUE - - B

SERIES ANTENNA

PART NUMBER 201910575-10

Associated PCB Stackup:

The PCB is a 4 copper layers stackup, 0.63mm thick with plated thru holes.

	Material (For information Only)	ϵ_r (1)	Tan Δ	Thickness (μm) (2)
ENIG plating				4
Solder mask	Probimer 77	4 @ TBD		20
Copper 1				35
Core	IT 158	4.1 @ 5GHz	0.018 @ 5GHz	70 (3)
Copper 2				18
Prepreg	IT 158	4.1 @ 5GHz	0.018 @ 5GHz	319
Copper 3				18
Core	IT 158	4.1 @ 5GHz	0.018 @ 5GHz	70 (3)
Copper 4				35
Solder mask	Probimer 77	4 @ TBD		20
ENIG plating				4
TOTAL :				613

(3): Works with 81 μ Nominal Thickness Cores

Table 1: PCB Stackup

PCB characteristics

- Copper :
 - Min track width / spacing : 0.12mm
 - Tolerance : +/- 40 μm
- Vias :
 - Plated Thru Holes
 - Drill diameter : 0.2mm
 - Pad diameter : 0.45mm (inner and outer layers)
 - Plugged vias (to control the quantity of solder paste on the pads surface)
 - Via stitching : 0.5mm pitch typ. (0.7mm max)
- Surface finish: ENIG plating without solder mask is suggested on the transmission lines to minimize losses between chips and antennas.

The antennas are 1 port component (pad location is indicated below, p. 5 and 8). The pad is where the track coming from the chip will be attached.

If required by your PCB design tool, you might need to add 1 port for each ground via (in the pads) so they are part of the component footprint.