

i50 Antenna Information

Antenna is built from a reference design (Texas Instruments Design Note DN004)
From DN004:



Design Note DN004

3 Description of the folded dipole design

Since the impedance of the folded dipole is matched directly to the impedance of the radio no external matching components are needed. Dependent on which regulatory limits the application should comply with, two filtering capacitors might however be needed. To ensure compliance with ETSI requirements for receiver spurious emission, C121 and C131 in Figure 3 must be included. The optimum value of C121 and C131 depends on which ETSI standard to comply with. For EN 300 440, 2.2 pF is recommended and 1.5 pF is recommended for EN 300 328. The required filtering for compliance with FCC is dependent on which regulatory part to meet, part 15.247 or 15.249, and the duty cycle of the transmitted signal. Including C121 and C131 reduces the output power but ensures lower harmonic emission in TX and reduced VCO leakage in RX. Typically a board with two 1.5 pF capacitors will have 2 dB less output power than a board without C121 and C131.

3.1 Implementation of the folded dipole

It is important to make an exact copy of the antenna dimensions to obtain optimum performance. The easiest approach to implement the antenna in a PCB CAD tool is to import the antenna layout from either a gerber or DXF file. Such files are included in the reference design, "CC2500EM_FD_Reference_Design_1_1.zip" available at www.ti.com/lpw. The gerber file is called "CC25xx_Folded_Dipole.spl" and the DXF file is called "CC25xx_Folded_Dipole.dxf". If the antenna is implemented on a PCB that is wider than the antenna it is important to avoid placing component or having a ground plane close to the end points of the antenna. If the CAD tool being used doesn't support import of gerber or DXF files, Figure 2 and Table 1 can be used. Since the pinout of CC251x differs from CC2500 it might be necessary to make a slight modification on the routing of the antenna feed lines. If this is done it is important to keep the same distance between the chip and the antenna.

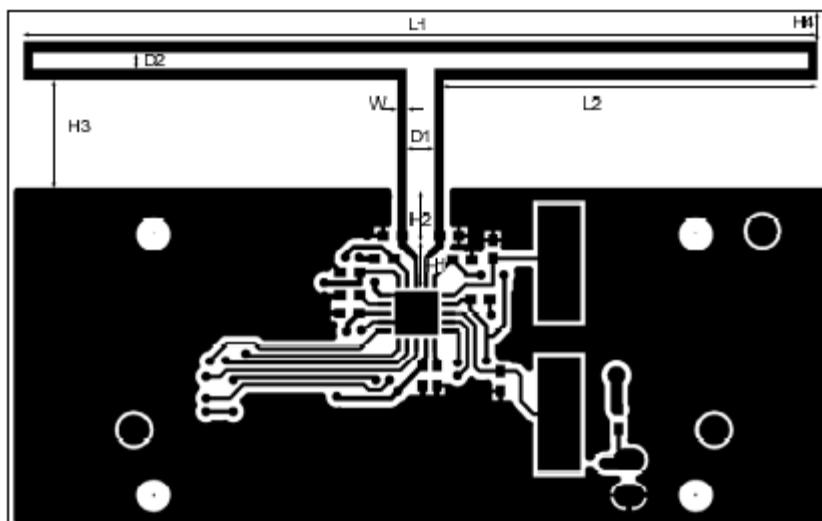


Figure 2: Antenna dimension.

L1	44.7 mm	H1	2.4 mm
L2	21.0 mm	H2	3.1 mm
D1	1.5 mm	H3	6.0 mm
D2	0.9 mm	H4	2.8 mm

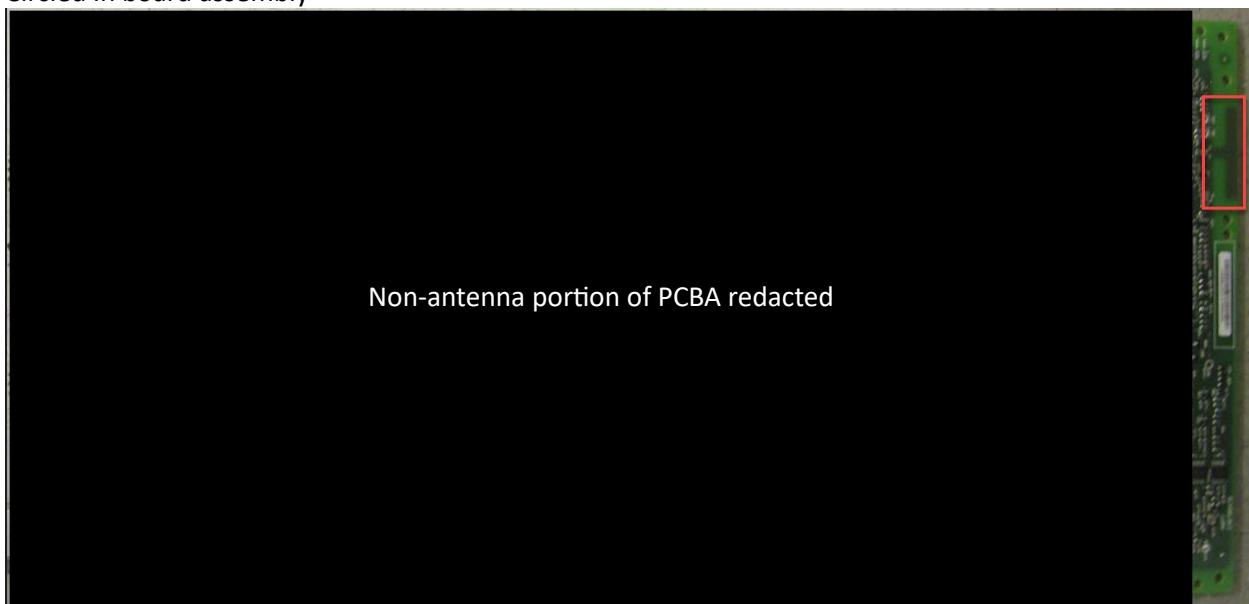
Table 1: Antenna dimension.

i50 Antenna Photos:

Close up



Circled in board assembly



Non-antenna portion of PCBA redacted