BOOSXTL-CC3120MOD

Very confusing Antenna design !

Summary

The recommended chip-antenna is a Taiyo Yuden AH316M245001-T. This antenna has a recommended opening with no ground plane under the opening and chip. The Certified reference design for the BOOSXTL-CC3120MOD has a continuous ground plane under the chip-antenna and opening.

The Coplanar Waveguide design recommendations are also not very clear. The impedance should be 50 ohm but using the data in the tables on page 44 in the CC3120MOD datasheet (SWRS205A –MARCH 2017–REVISED JUNE 2017) you get 61.5 ohms and 56.3 ohms respectively.

Questions

See documentation below for information.

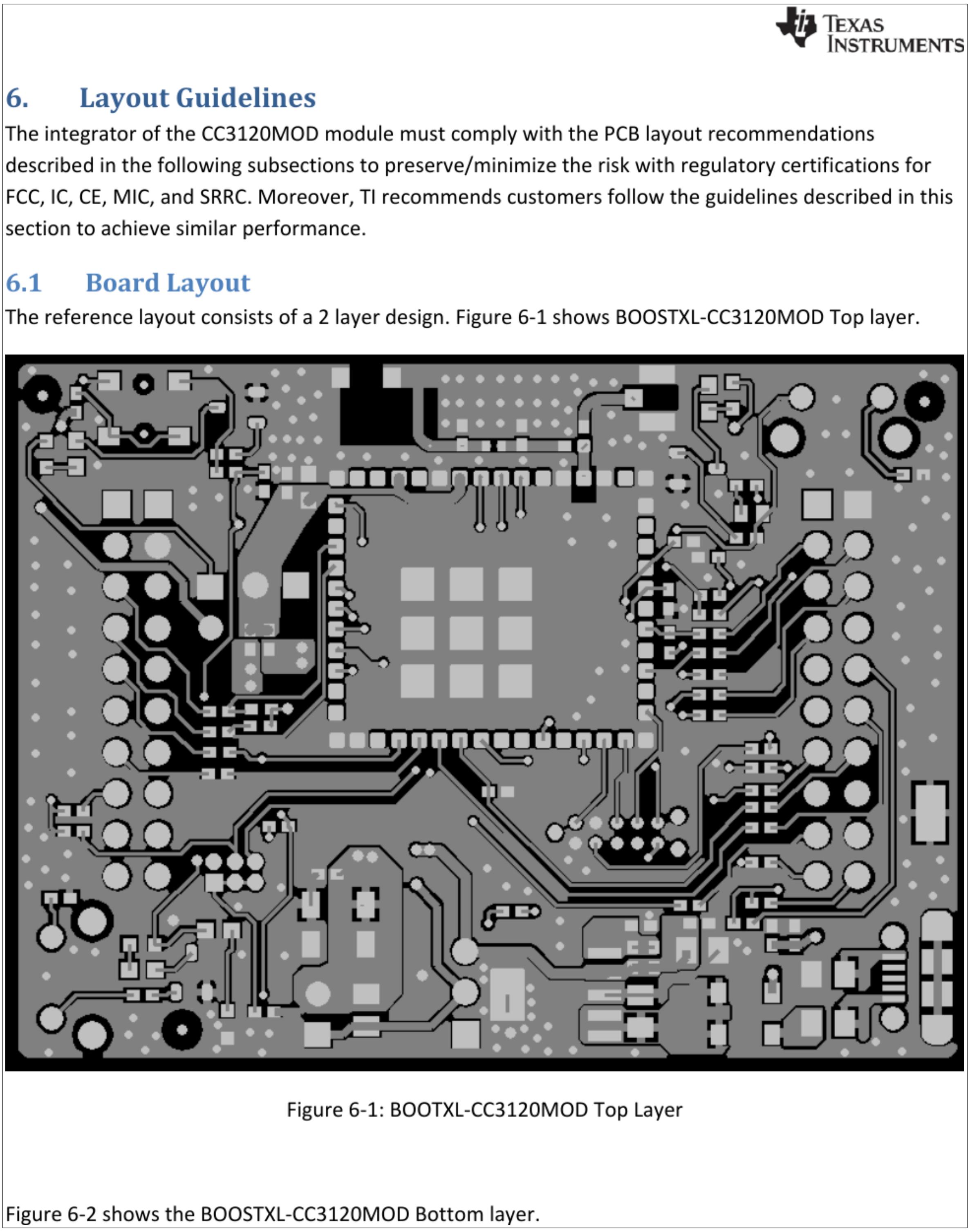
- What is the best way follow the 'Reference Design" ? Implementing according to the TI layout or just skipping what you have done
  and follow TI recommendations in the texts and the Taiyo Yuden Antenna Design Guide ?
  - Will this violate the Reference design?
  - Does the Reference design with a solid ground plane under the antenna actually work?
- Why do the TI Coplanar Waveguide recommendations differ significantly from 50 Ohms ?
  - Are our calculations not correct?
  - Are the TI deviations from 50 ohm insignificant?
  - Have TI tweeked the design to work with a groundplane under the antenna?

Links

- http://www.ti.com/tool/BOOSTXL-CC3120MOD
- Chemandy - Coplanar Waveguide Calculator
- fccid.io CC3120
- Japansk CC3220, "Void space on all layers"
- Japansk CC3100MOD
- Taiyo Yuden
- AH316M245001-T, **here** is a good link to the Antenna Design Guide (The Taido Yuden homepage is very slow)

Findings

Looking at the top and bottom layer for the reference design, taken from this link. The same information is found downloading and unzipping the swrc339 (BOOSXTL-CC3120MOD_Design_Files), looking through the Layers PDF or the Gerbers (See under Design Files here).



TEXAS INSTRUMENTS

6. Layout Guidelines

The integrator of the CC3120MOD module must comply with the PCB layout recommendations described in the following subsections to preserve/minimize the risk with regulatory certifications for FCC, IC, CE, MIC, and SRRC. Moreover, TI recommends customers follow the guidelines described in this section to achieve similar performance.

6.1 Board Layout

The reference layout consists of a 2 layer design. Figure 6-1 shows BOOSTXL-CC3120MOD Top layer.

Figure 6-1: BOOTXL-CC3120MOD Top Layer

Figure 6-2 shows the BOOSTXL-CC3120MOD Bottom layer.

PICTURE 1

Cut Out here !?

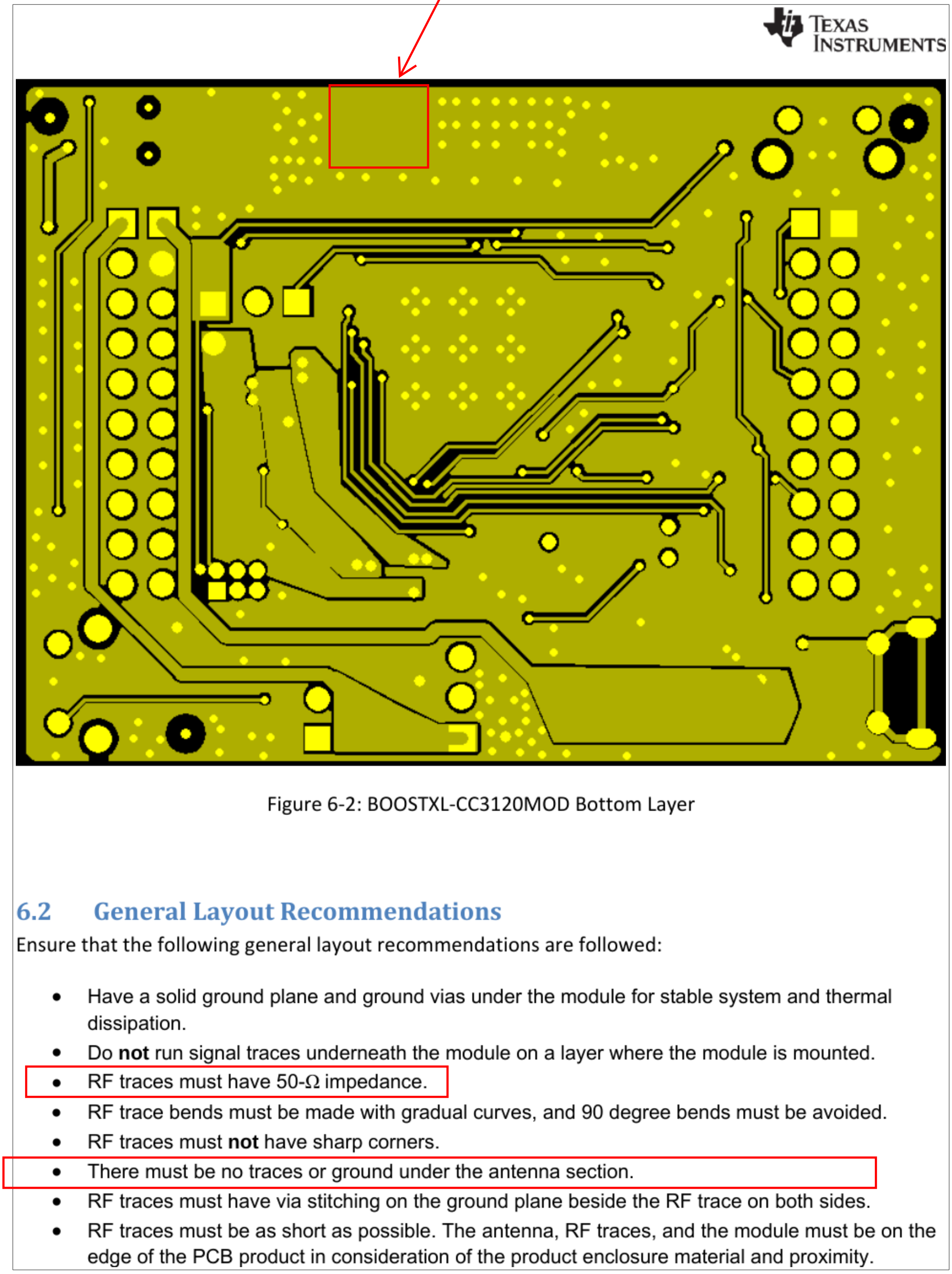


Figure 6-2: BOOSTXL-CC3120MOD Bottom Layer

6.2 General Layout Recommendations

Ensure that the following general layout recommendations are followed:

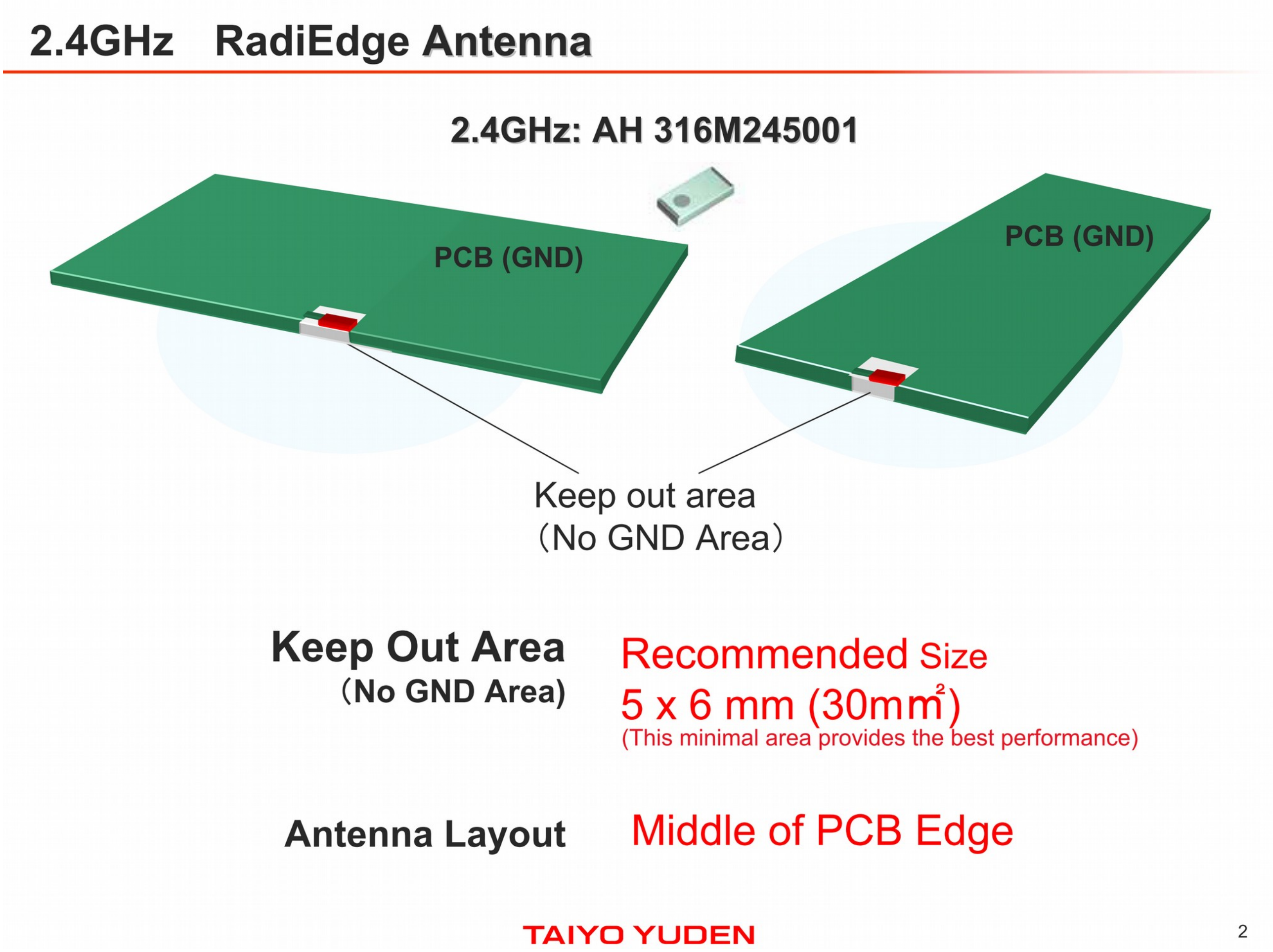
- Have a solid ground plane and ground vias under the module for stable system and thermal dissipation.
- Do **not** run signal traces underneath the module on a layer where the module is mounted.
- RF traces must have 50-Ω impedance.
- RF trace bends must be made with gradual curves, and 90 degree bends must be avoided.
- RF traces must **not** have sharp corners.
- There must be no traces or ground under the antenna section.
- RF traces must have via stitching on the ground plane beside the RF trace on both sides.
- RF traces must be as short as possible. The antenna, RF traces, and the module must be on the edge of the PCB product in consideration of the product enclosure material and proximity.

PICTURE 2

What sticks out here is:

- “RF traces must have 50 Ohm impedance”
- “There must be no traces or ground under the antenna section”, the above design contradicts this !

From Taiyo Yuden Antenna Design Guide.



2.4GHz RadiEdge Antenna

2.4GHz: AH 316M245001

Keep out area
（No GND Area）

Keep Out Area
（No GND Area）

Recommended Size
5 x 6 mm (30mm²)
(This minimal area provides the best performance)

Antenna Layout Middle of PCB Edge

TAIYO YUDEN

PICTURE 3

CPW

From CC3120MOD (SWRS205A –MARCH 2017–REVISED JUNE 2017), page 44.

Putting in the figures in the CPW-calculator (see Links above) you get the following values, se Picture 4 below.

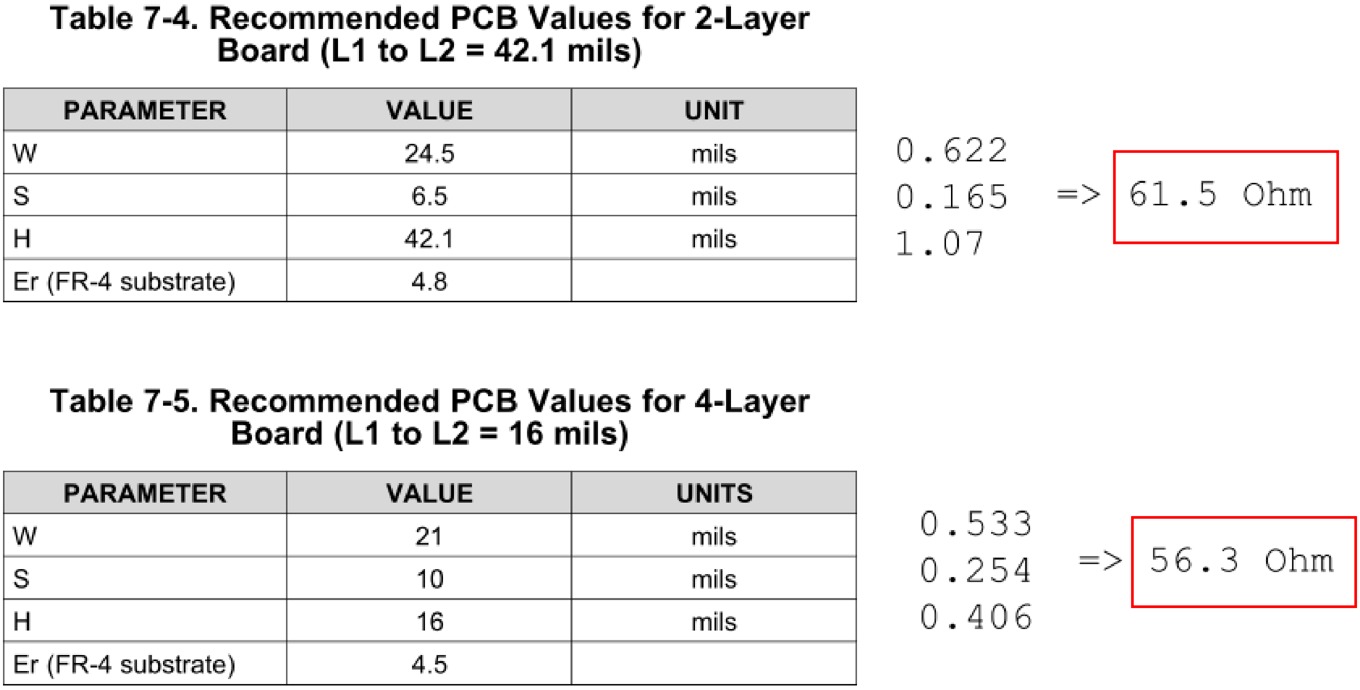


Table 7-4. Recommended PCB Values for 2-Layer Board (L1 to L2 = 42.1 mils)

PARAMETER	VALUE	UNIT
W	24.5	mils
S	6.5	mils
H	42.1	mils
Er (FR-4 substrate)	4.8	

0.622
0.165 => 61.5 Ohm
1.07

Table 7-5. Recommended PCB Values for 4-Layer Board (L1 to L2 = 16 mils)

PARAMETER	VALUE	UNITS
W	21	mils
S	10	mils
H	16	mils
Er (FR-4 substrate)	4.5	

0.533
0.254 => 56.3 Ohm
0.406

PICTURE 4

)) Kind regards, Per