Pulse Antenna Solutions

Basics of Pulse Ceramic Chip Antenna

Pulse | 2015

Topics

- Antenna Design Method
- PIFA Antenna Basics
- Considerations of Antenna Implementation on PCB

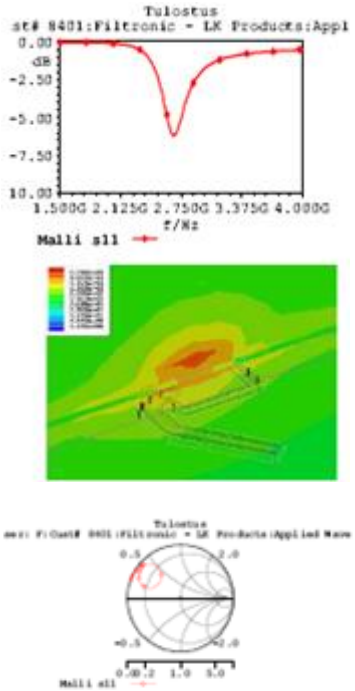
Pulse Electronics

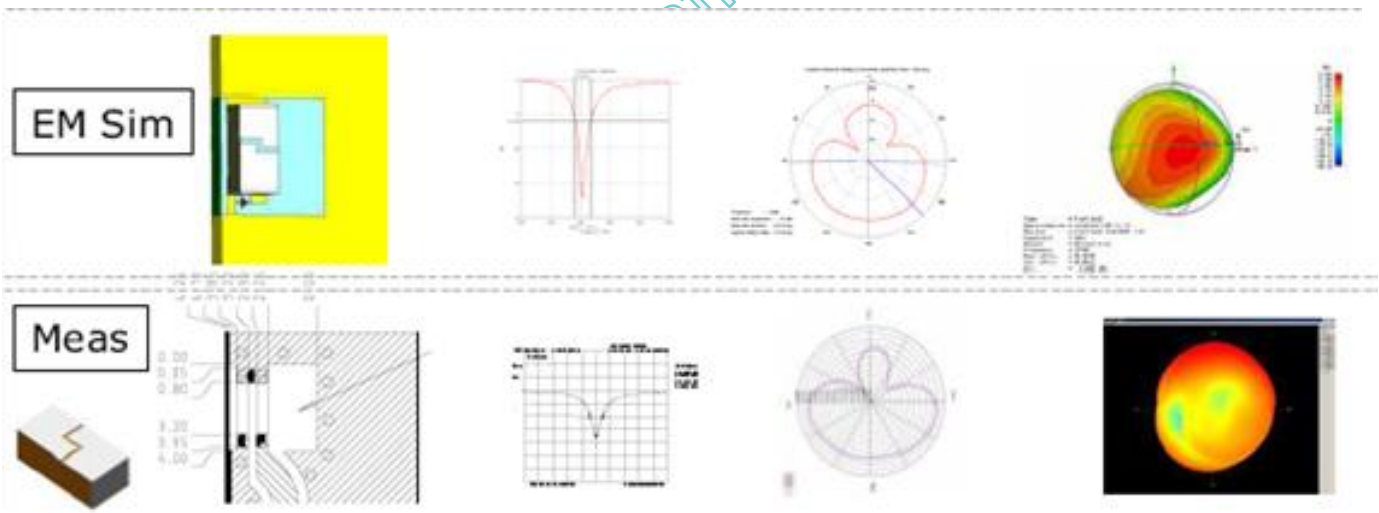
Antenna Design Method

Pulse Electronics

Simulation / Design Tools

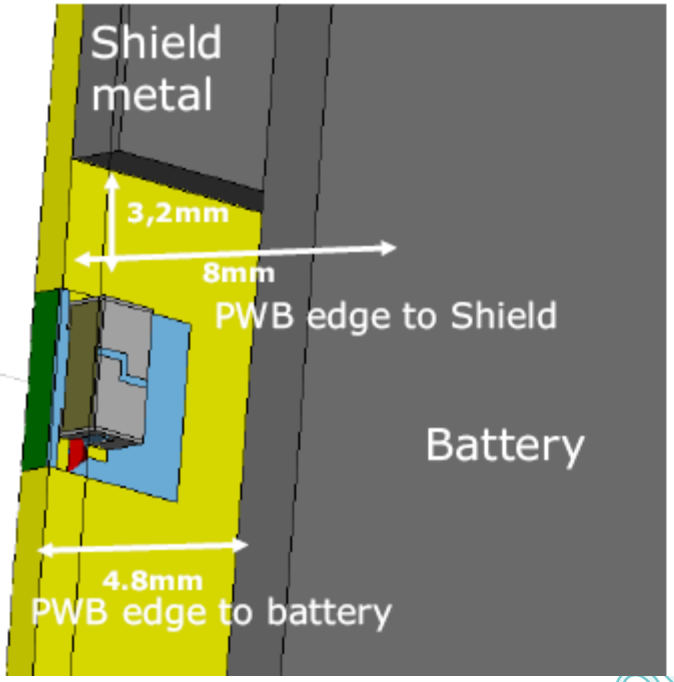
- Mechanical Design:
  - Catia, I-DEAS, PRO-E, SolidWorks
- Electrical Design:
  - CST MWS, IE3D, AWR Aplac, AWR MWO, Ensemble, HFSS Ansoft Designer
    - ➢ Structural Simulation
    - ➢ Tolerance Analysis
    - ➢ Material Studies

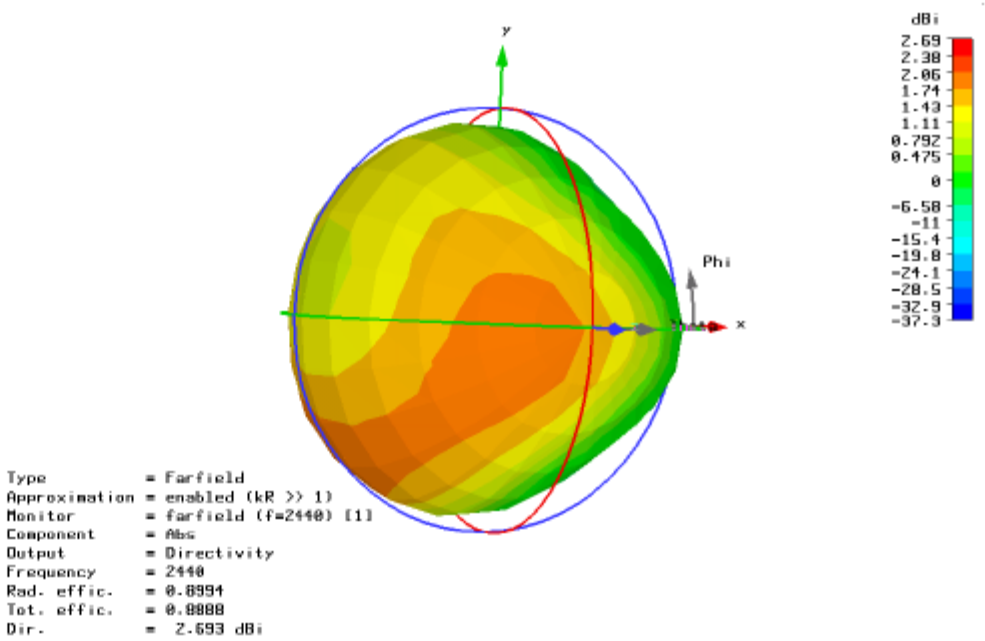




Pulse Electronics

Example of EM simulation model

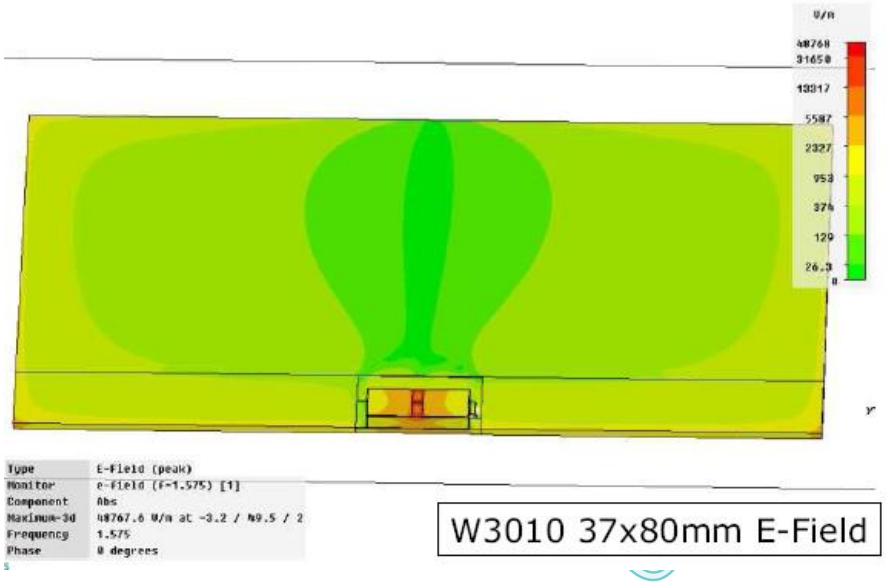




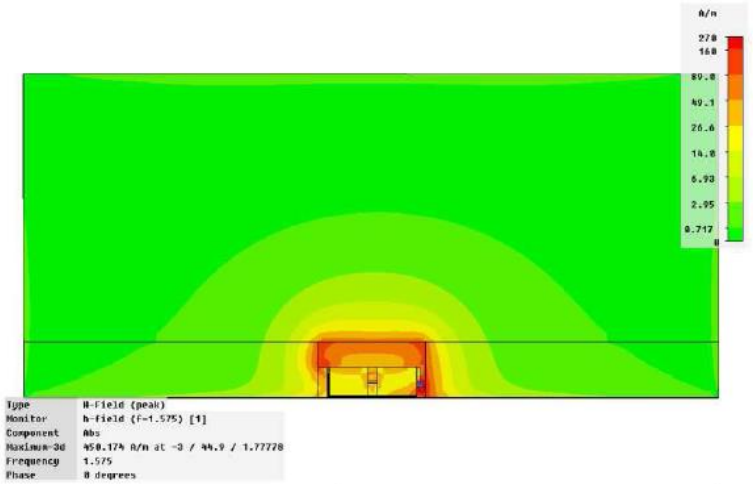
- W3008 model build using CST MWS

Pulse Electronics

Simulated EM fields (Example: W3010)



W3010 37x80mm E-Field



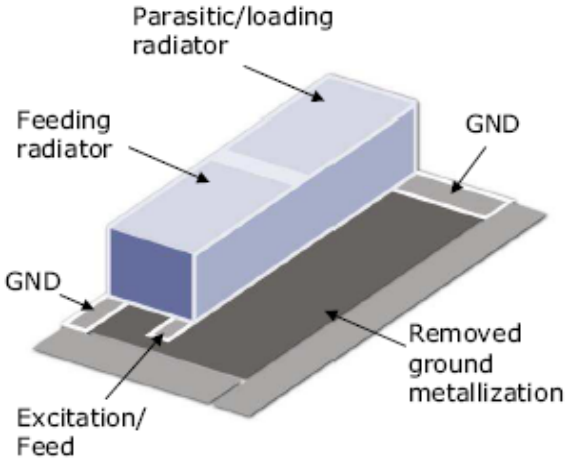
W3010 37x80mm H-Field

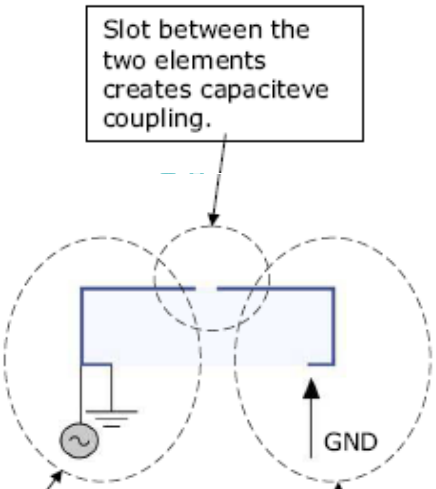
Whole PCB is “hot”

PIFA Antenna Theory (Ceramic PIFA)

Pulse Electronics

Loaded PIFA Structure





Slot between the two elements creates capaciteve coupling.

First element forms a PIFA type radiator having Feed and Ground contacts

Second element is electrically connected to first through a capacitive coupling.

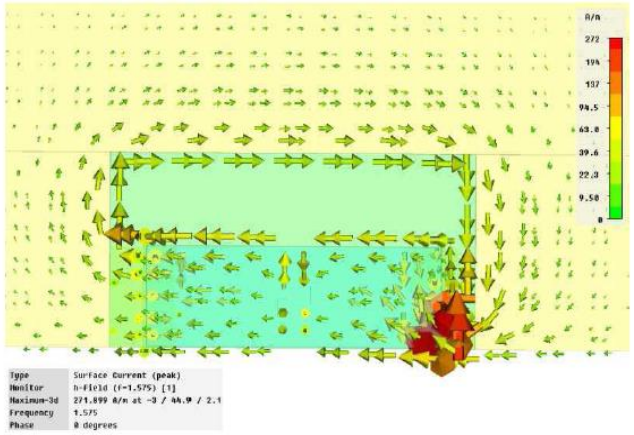
Second elements tunes resonance frequency downwards enabling use of physically small antenna element.

Stucrure is called **"Loaded PIFA"** due to frequency tuning effect of second element.

Pulse Electronics

Function of Ground Clearance area / layout

Simulated Surface currents
(Example: W3010)



GC area dimensions determine impedance bandwidth. Larger area results wider BW.

GC area dimensions also determines the currents return path distance from grounded element back to feed. Bigger distance results lower frequency.

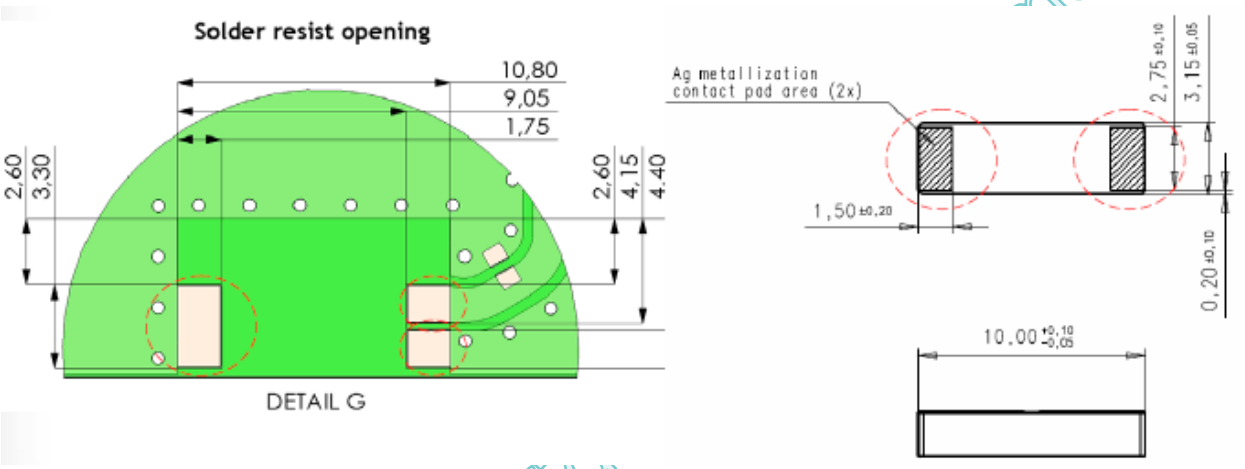
Matching circuits are used for final impedance optimization.

First element is exited by an feed electrode.
With GND-contact this element forms a PIFA type radiator.

Second grounded element is coupled to feeding element via narrow gap in metallized surface. Amount of coupling determines how much the antenna resonance frequency is loaded.

Pulse Electronics

Why three pads on board and two on antenna?



- Basic antenna is PIFA type; Feed and GND are needed for first electrode
- Antenna pad short circuits the two pads on board
- Board pads are normally kept separate to make impedance matching more stable and straight forward

Pulse Electronics

Benefits of “Loaded PIFA”

- Can achieve extremely good efficiency number when properly implemented, 90%
- Creates close to omni-directional 3D radiation pattern
  - Above mentioned features are due to combination of ceramic antenna element and board resonance radiation
- High immunity of frequency detuning due to user tissue (body, hand, head) and surrounding mechanics

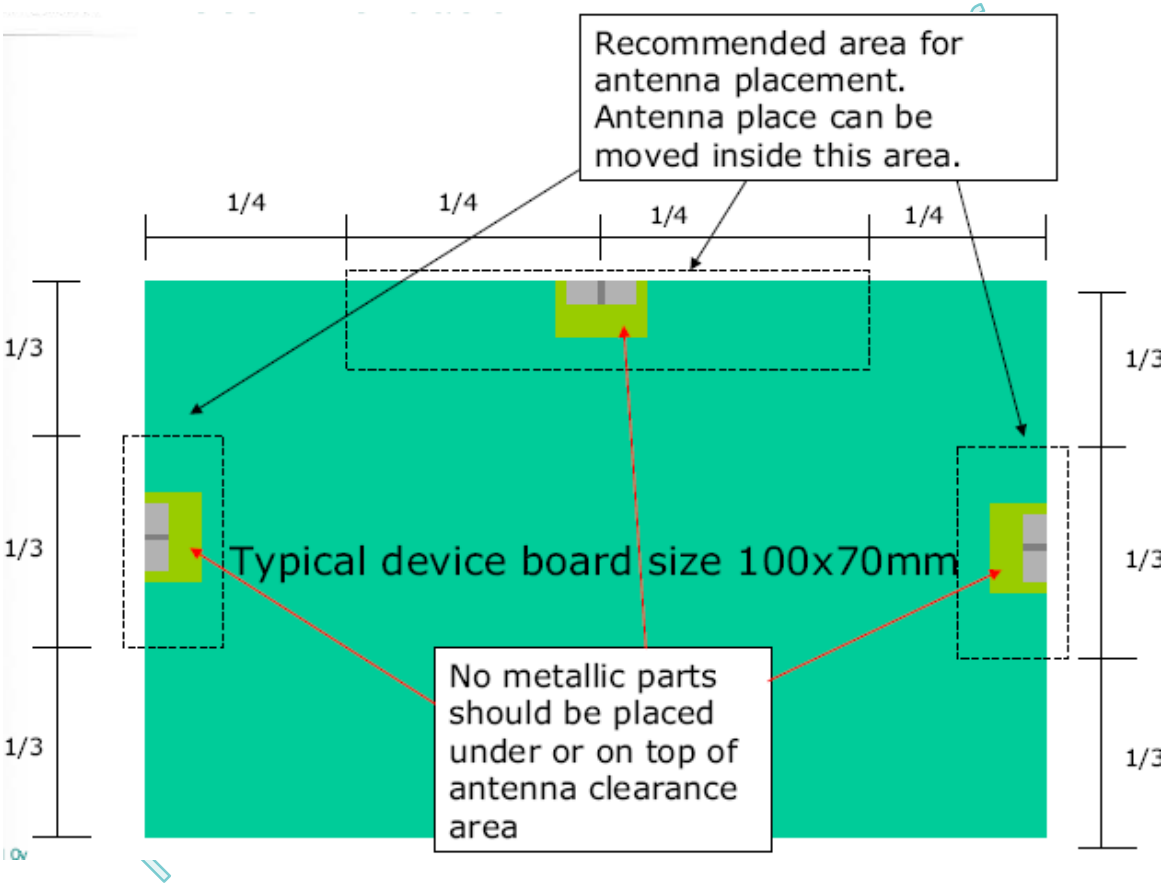
Considerations of Antenna Implementation on PCB

Pulse Electronics

PCB effects

- Loaded PIFA antenna is short circuited to PCB ground
- Antenna element electrical length is λ/4
- PCB has major effect in overall antenna performance figures!
  - PCB electrical length (examples shown later in this presentation)
  - Antenna position on board (examples shown later in this presentation)
  - Grounding points on antenna layout
  - PCB layout and size affects on antenna:
    - Frequency
    - Bandwidth
    - Feed impedance
    - Total radiation performance

Antenna Placement Recommendations



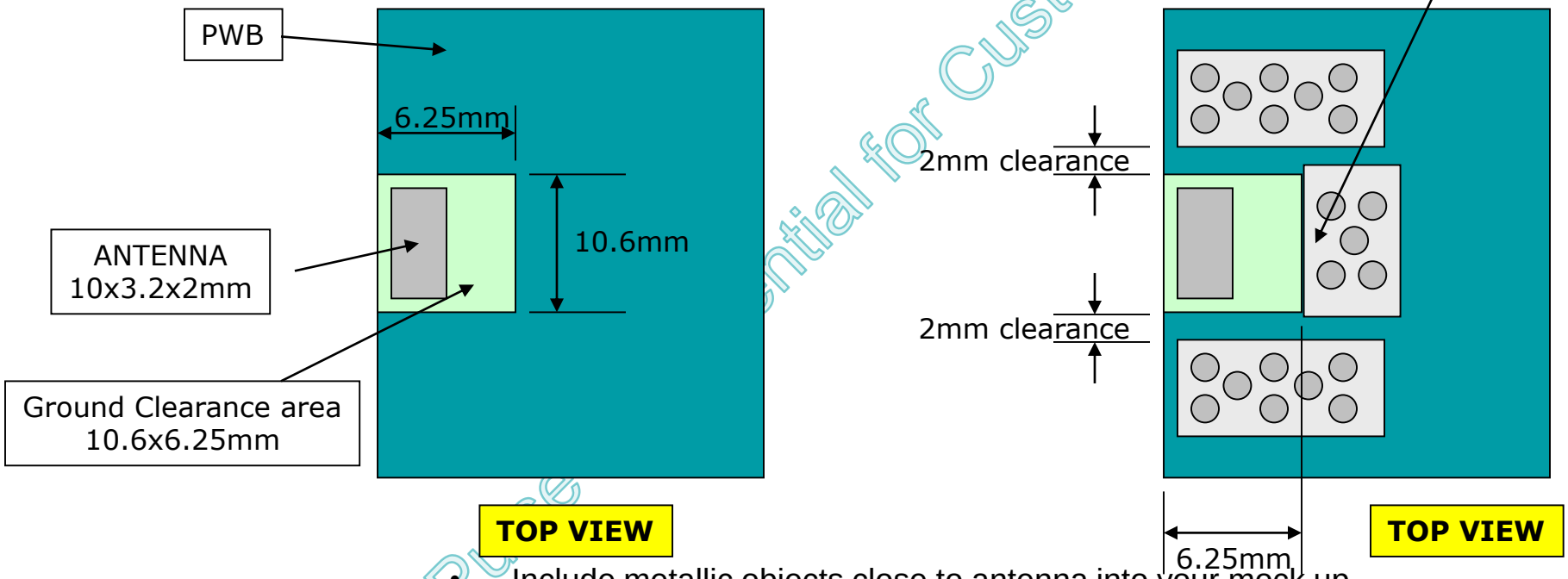
- Fractions represent wavelengths

Recommended metal object guard distances for ground clearance (Example: PIFA type W3010 chip antenna)

Same design rules apply to both sides of the PWB, top and bottom

Height of metal object has effect on antenna performance. Over 2mm heigh metallic parts decrade antenna performance if placed close to antenna.

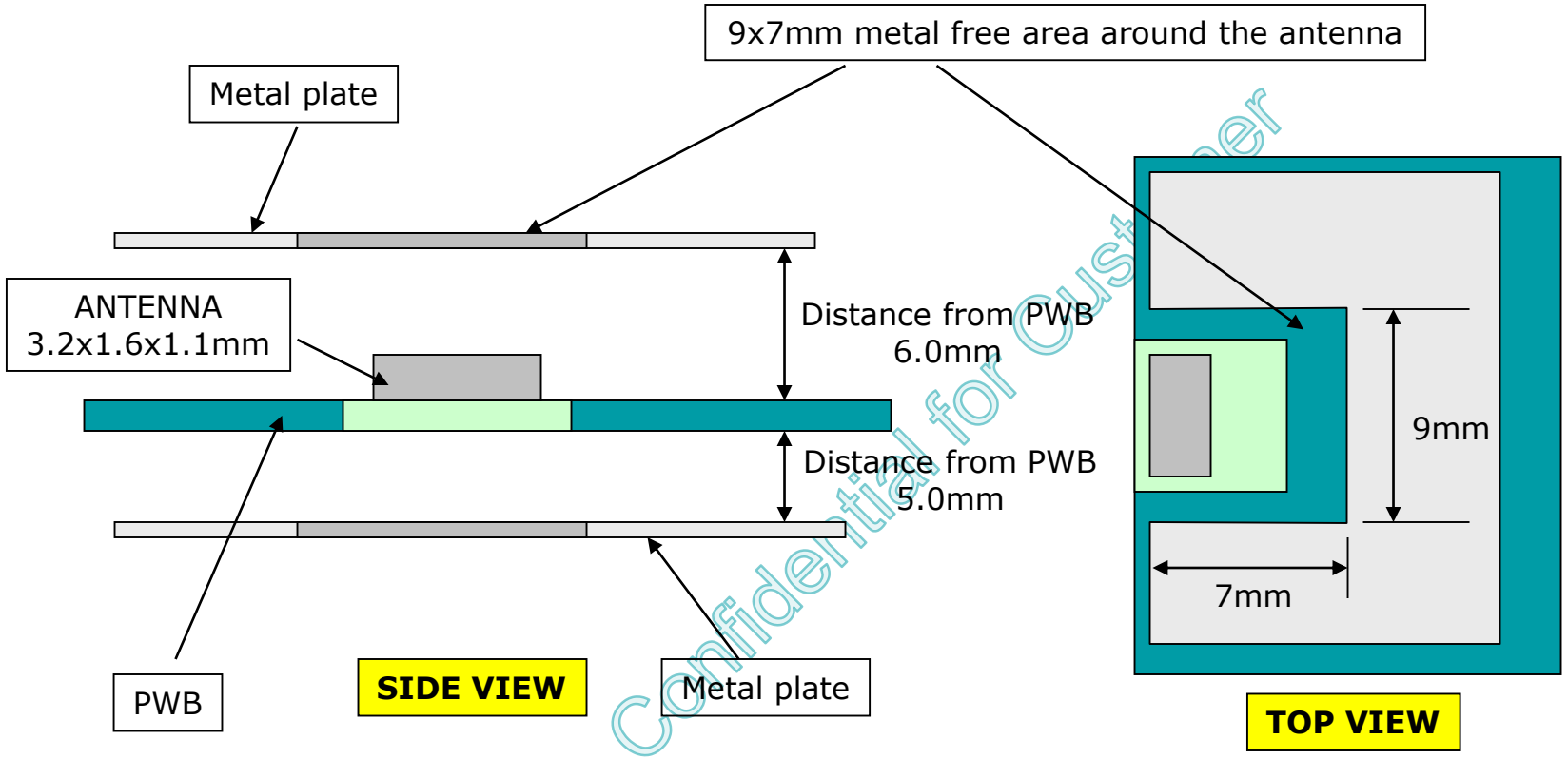
Metal objects like shield cans can be placed on the edge of the antenna ground clearance area



- Include metallic objects close to antenna into your mock up testing as early as possible!
- Common Metallic objects : Connector, Switches, Wibra, Microphone, Shield cans, Etc.

Pulse Electronics

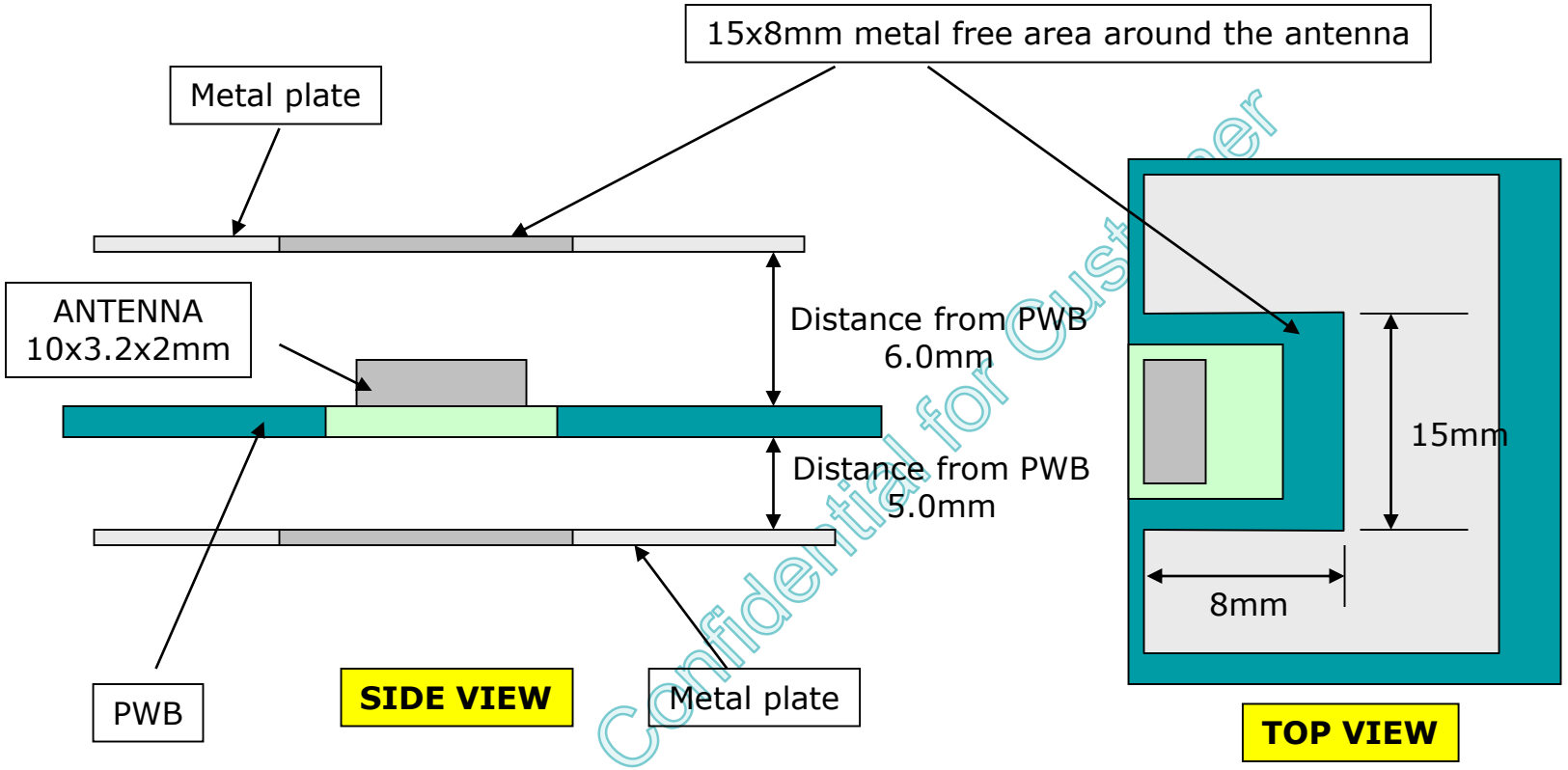
Recommended metal object guard distances for ground clearance (Example: PIFA type W3011 chip antenna)



This information of metallic plate/cover guard distance is indicative only and should be concidered as minimum keep out distances. The best radiating performance is allways achieved when antenna is in as ”free space” condition as possible. It is recommended to use low dielelctric low loss plastic covers.

Pulse Electronics

Recommended metal object guard distances for ground clearance (Example: PIFA type W3010 chip antenna)

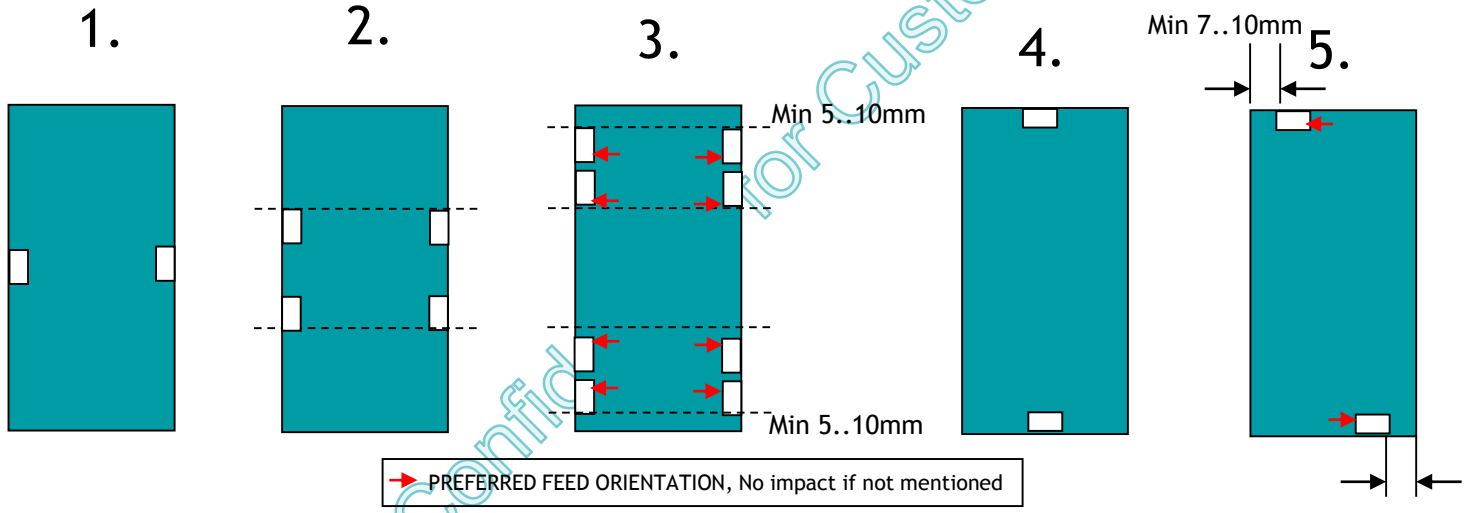


This information of metallic plate/cover guard distance is indicative only and should be concidered as minimum keep out distances. The best radiating performance is allways achieved when antenna is in as ”free space” condition as possible. It is recommended to use low dielelctric low loss plastic covers.

Pulse Electronics

“OK to use” antenna position on PWB (Example: W3010 GPS)

- Typical board side 35..45 x 35..110mm
- Each used position may require external matching components and ground clearance area modifications for finetuning the impedance matching and center frequency tuning



Left or right side center: Best position

Midle third of the PWB: Ok position without significant change in performance

Minimum 5 to 10mm from top or bottom edge: Position with about 0.5..1dB drop in performance

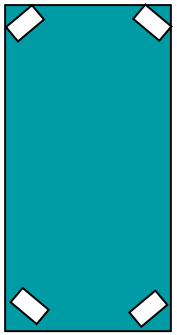
Top or bottom end of PWB: Position with about 0.5 to 1dB drop in performance, depends on bord width, wider is better

Top or bottom end corner of PWB: Position with about 0.5 to 1dB drop in performance, depends on bord width, wider is better

“NOT OK to use” antenna position on PWB (Example: W3010 GPS)

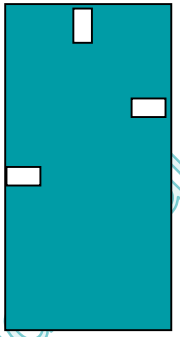
- Below mentioned positions will result to poor performance of the antenna

6.



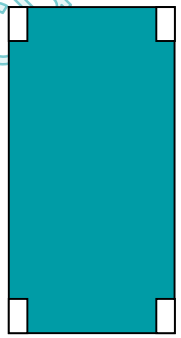
Top or bottom corner 45deg angle:
NOT to use

7.



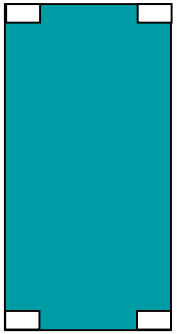
”Sideways” in any position:
NOT to use

8.



Directly on corner vertically:
NOT to use

9.



Directly on corner horizontally:
NOT to use

Pulse Electronics

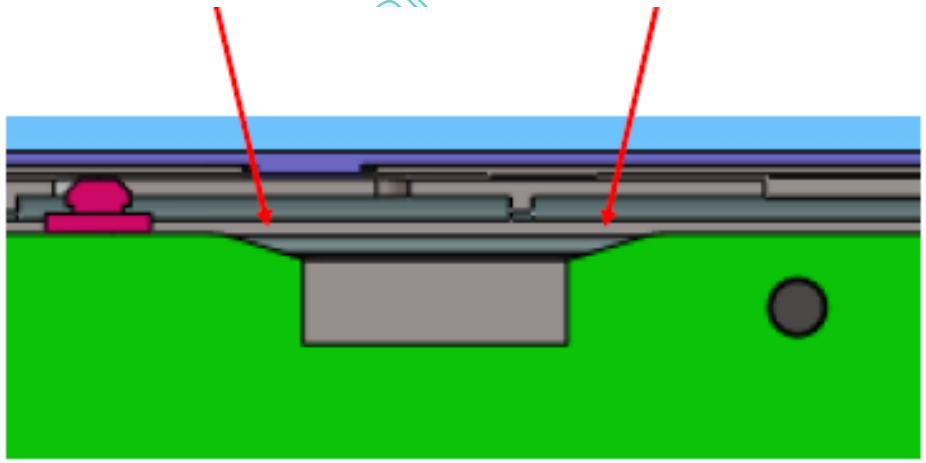
Do's and Dont's

- Do
  - Use Pulse footprint recommendation as baseline
  - Place enough grounding vias on the edges of the clearance area. Route vias through all layers in the board.
  - Clear the metal away from all the layers of the board
  - Make needed matching and tuning by clearance area changes and external mathing components
  - Use plastic covers

- Do NOT
  - Place antenna directly in the corner of the board
  - Place any components or traces on the antenna clearance area (all layers)
  - Place metallic covers on top or below the antenna and clearance area

Board edge

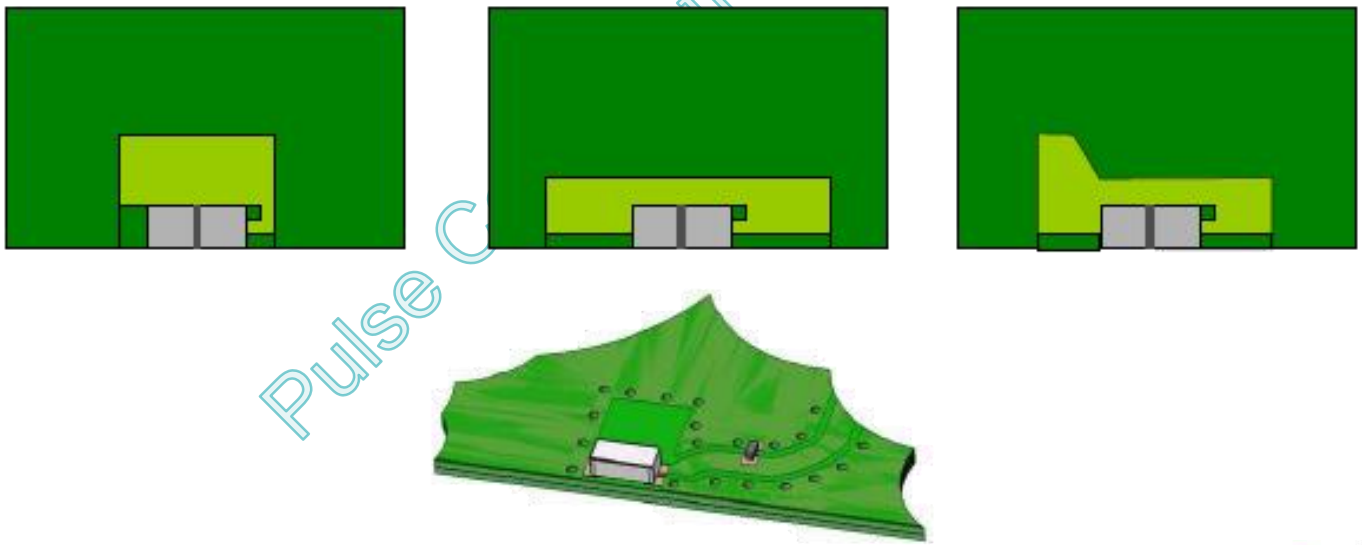
- It is ok to have antenna moved couple millimeters inwards to PCB
- Ground copper on antenna corners should be chamfered to minimize effect on performance



Ground Clearance area shape

- GC-area does not need to be square shaped. Square shape is presented in Pulse apps notes as a standard starting point for the layout work.

- Arbitrary form GC-areas can be used as long as total area and current return path distance is optimized to give correct resonant frequency and BW



Pulse Electronics

Consistency of Ground

- Antenna sees the whole PCB as ground plane
- Overall board dimensions determine the PCB electrical length
- Especially around the GC area ground pour needs to be solid
- Ground pour does not need to be continued on same layer over the whole board. Several layers can be connected together with via holes

Pulse Electronics

Multi-layer board considerations

- Again most critical point on multi-layer layout is the GC area surroundings
- All layers around the GC area must be connected together to avoid signal coupling/leaking into gaps between the layers
- Poorly grounded GC are also causes problems in impedance matching and frequency control
- DC voltage layers can be left floating as long as metal of that layer does not overlap with the GC area

“THANK YOU”

Pulse Electronics