



APPENDIX J

: ANTENNA SPECIFICATION



1. Introduction

Bluetooth chip antenna is a component which should be used with matching circuit composed of L or C.

This means that you have to match the antenna with your main board to get the best performance.

When you get a matching process with L or C, center frequency would be getting down to 2.45 GHz. We offer several models which have different size and center frequency, therefore you can choose the most proper one according to your board condition.

Part No.	Fc (GHz)	Gain (max. dBi)	Size (mm)	Application
ALA321C2	2.7	0	3.2x1.6x1	Bluetooth handset
ALA321C3	3.1			
ALA321C4	3.5			
ALA621C2	2.71	0	6x2x1	Bluetooth handset, Bluetooth device
ALA621C3	3.05			
ALA621C4	3.32			
ALA931C2	2.89	2	9x3x1	Bluetooth handset, Bluetooth device
ALA931C3	3.27			
ALA931C4	3.50			
ALA931C5	3.69			
ALA131C1	2.70	2.2	11x3x1	Bluetooth device, DMB device
ALA131C2	3.09			
ALA131C3	3.51			

* This frequency must be adjusted to 2.45GHz with matching circuit.

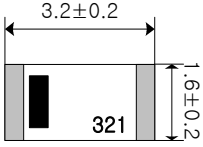
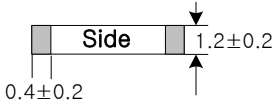
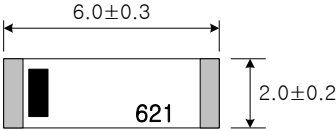
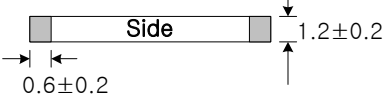
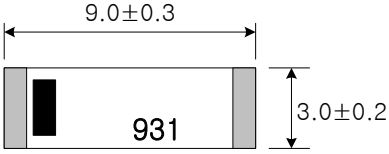
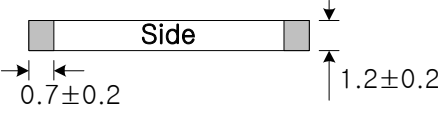
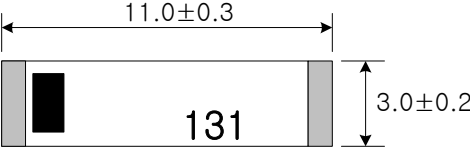
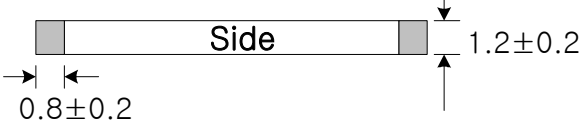
Antenna	321 series
Front view	
Side view	
Antenna	621 series
Front view	
Side view	
Antenna	931 series
Front view	
Side view	
Antenna	131 series
Front view	
Side view	

Fig. 1 Antenna dimension (Unit: mm)

2. Recommended position & environment

1) Antenna direction

Dummy side should be directed to the outside of PCB.

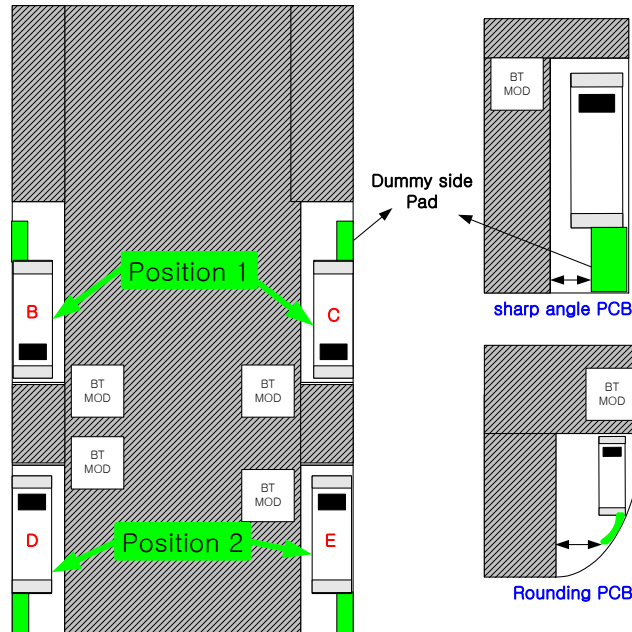


Fig. 2 Antenna position & dummy side

- Position 2 is better than Position 1, if both have the same 'no ground' size

2) Environment

Chip antenna should be placed far from conducting materials like, Battery, EMI shielding materials, Folder speaker, Metal case, etc.

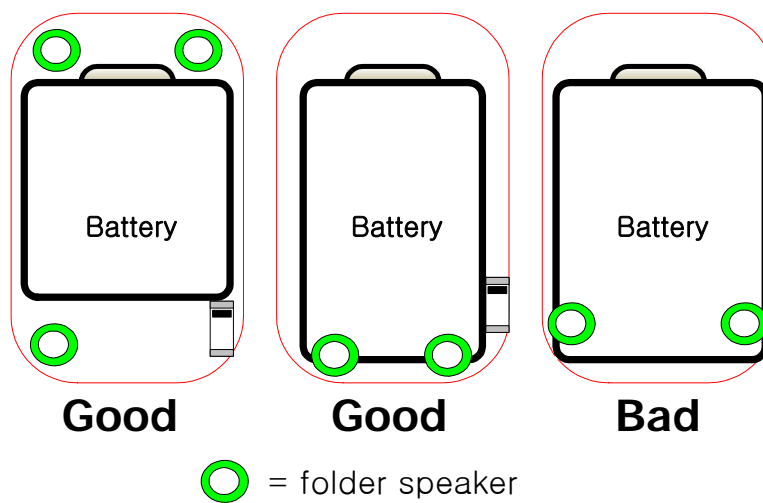


Fig. 3 Recommended Antenna position

3) No ground(clearance) size & antenna pad

The larger 'no ground area', the better performance, you should place antenna at least 1mm far from the ground area.

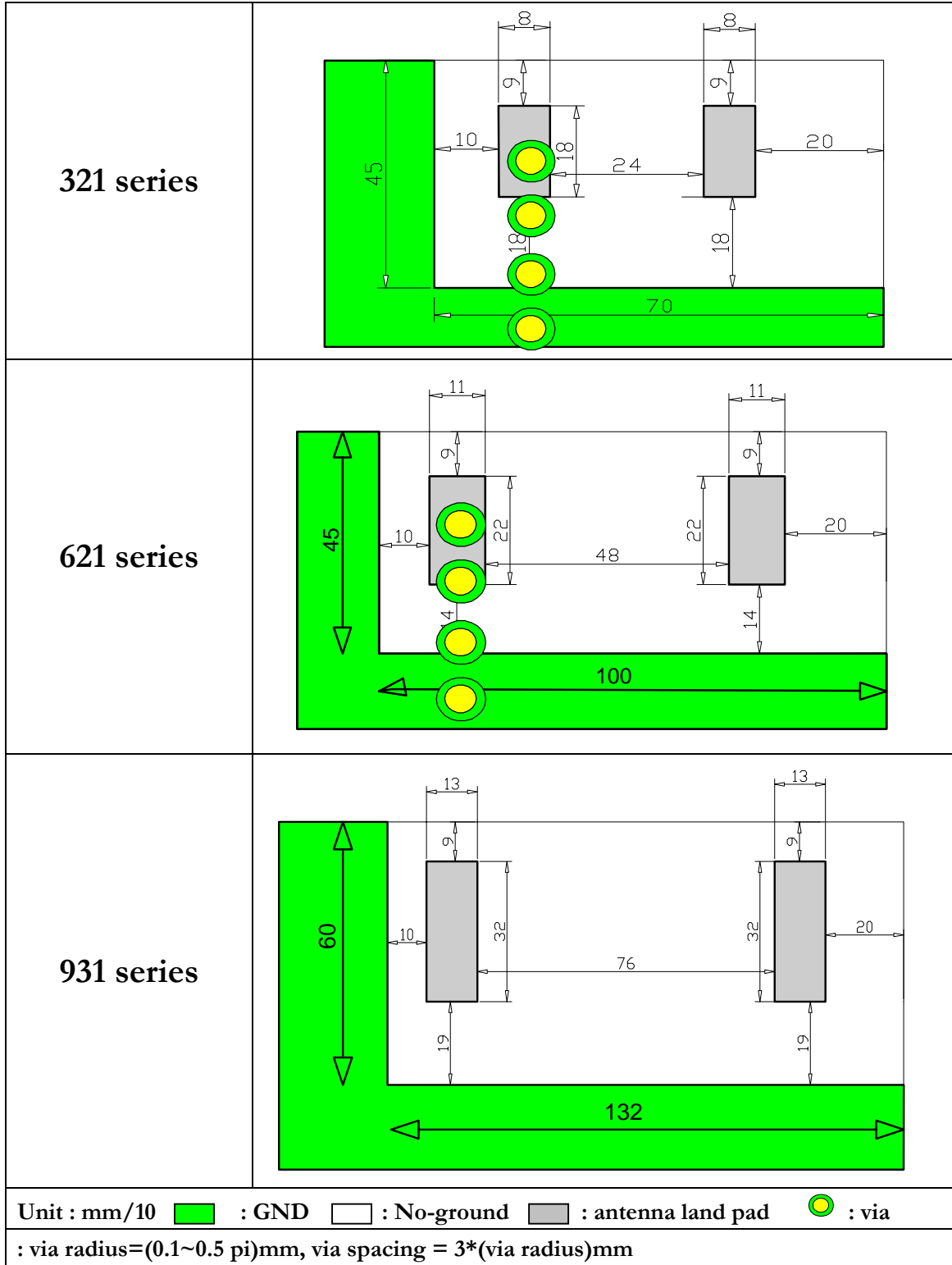


Fig. 4 Required No-ground area & antenna pad dimension

4) Required 'no ground size'

P/N	Antenna size (mm)	No ground size(min.) (mm)
ALA321C series	3.2 x 1.6	6 x 4
ALA621C series	6 x 2	10 x 4.5
ALA931C series	9 x 3	13.2 x 6

3. Matching circuit & component

Chip antenna should be matched with PCB.

Normally this process can be done with capacitor or inductor as follows,

Component	Description	Value
Capacitor	Series C	0.5 ~ 10 pF
	Through C	33, 100 pF
Inductor	Series L	1.0 ~ 6nH
	Shunt L	1.0 ~ 6nH

* Series : Connected between antenna and feeding line in series

Shunt : Connected between antenna and feeding line in parallel

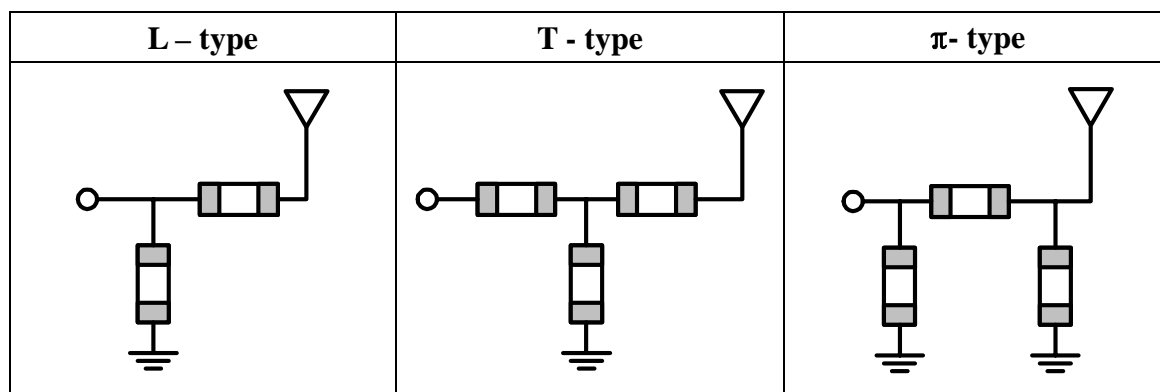


Fig. 5 Matching circuit & component

4. Measurement system

Test Result Value		
Measurement(matching)	S11 (HP8753ES network Analyzer)	dB
Measurement(radiation pattern)	S21 (HP8753ES network Analyzer)	dB _i
Radiation Test Condition		
Parameter	Condition	Unit
Chamber size	8*4*4	m
Temperature	21.5	°C
Humidity	55	%
Absorption rate	-50dB under (over 2,000MHz)	dB
System software	Midas (Orbit/FR)	Version 3.6

5. Matching process

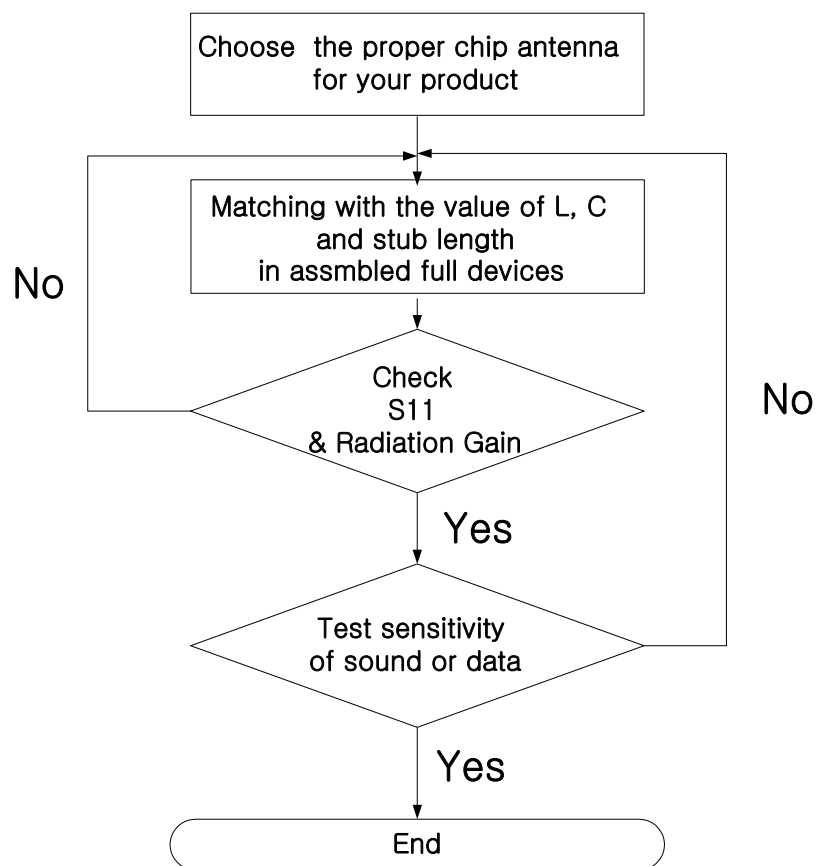
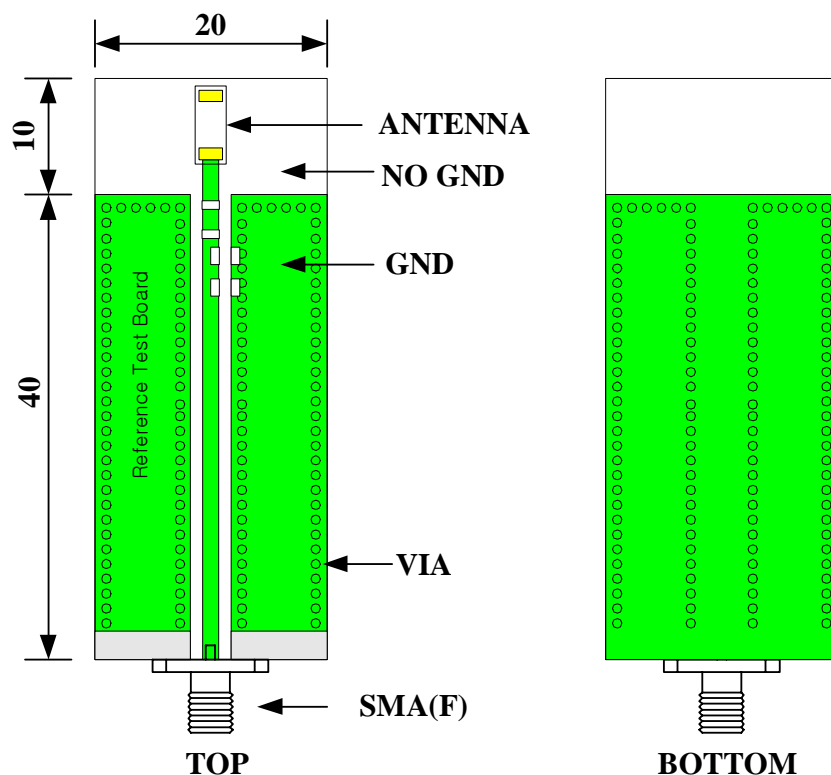


Fig. 6 Matching process diagram

6. Matching process example

1) Test PCB



2) Impedance matching value summary

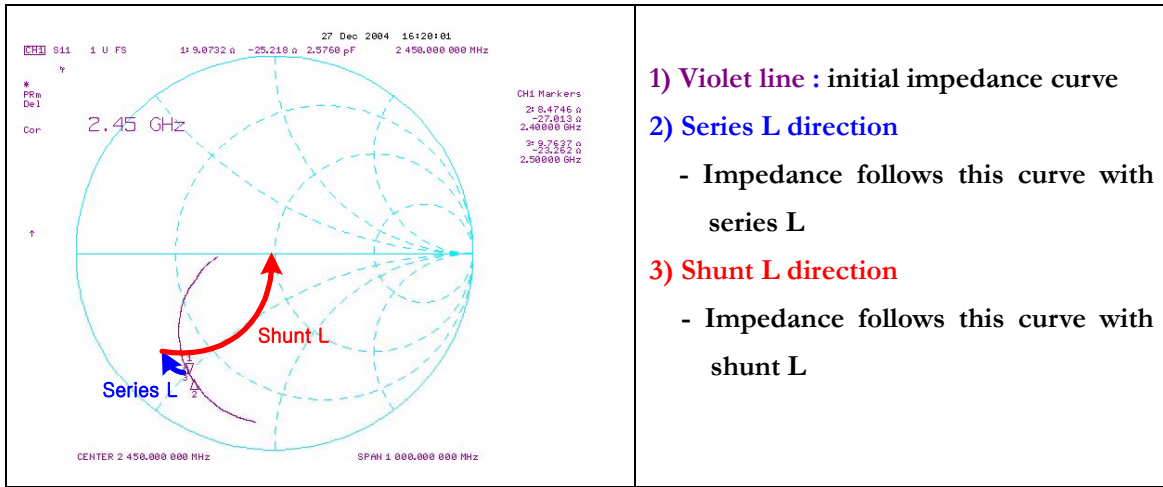
Model	f_0 [GHz] ¹⁾	Matching component value ²⁾	
		series	shunt
931C5	3.88GHz	2.2nH	2.2nH
931C4	3.63GHz	1.8nH	1.8nH
931C3	3.24GHz	1.2nH	1.8nH
621C4	3.35GHz	2.2nH	1.8nH
621C3	3.18GHz	-	2.2nH
321C4	3.65GHz	3.3nH	1.2nH
321C3	3.05GHz	2.7nH	1.2nH
321C2	2.81GHz	1.8nH	1.8nH

1) f_0 : Initial frequency of antenna before matching process

2) These components are needed to get an impedance matching at 2.45 GHz

But the value could be different according to the board condition.

2) Impedance adjustment process (Example : 931C3)



1) Violet line : initial impedance curve

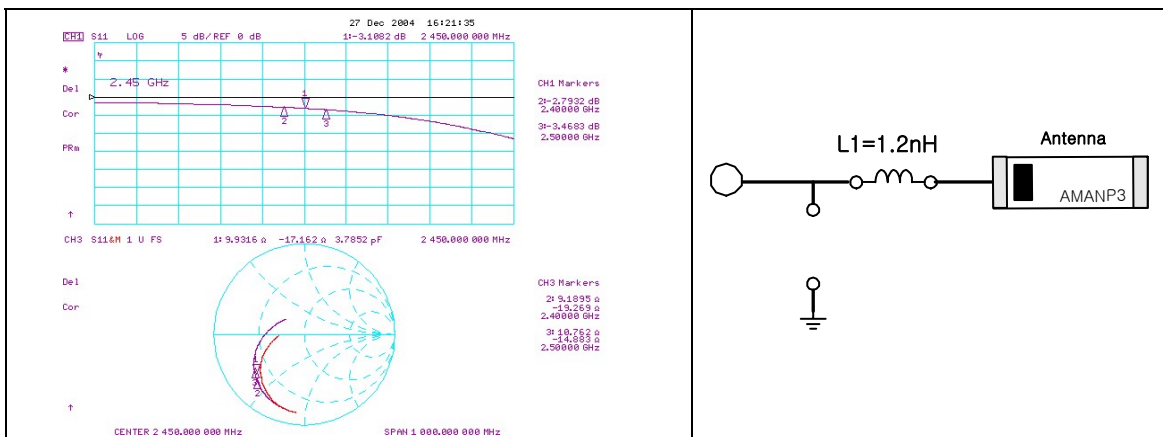
2) Series L direction

- Impedance follows this curve with series L

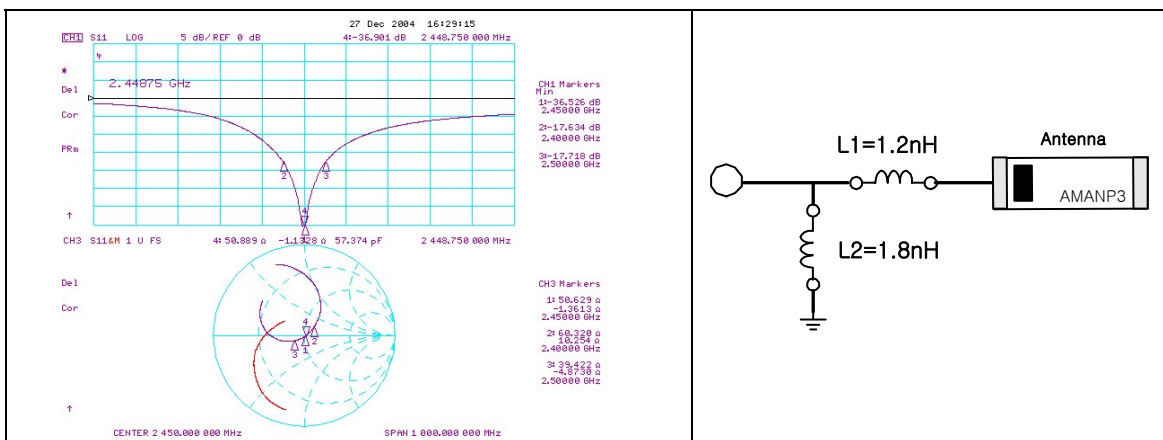
3) Shunt L direction

- Impedance follows this curve with shunt L

[Step 1 : 931C3 ideal matching process on Smith Chart]



[Step 2 : Lumped series L= 1.2nH]

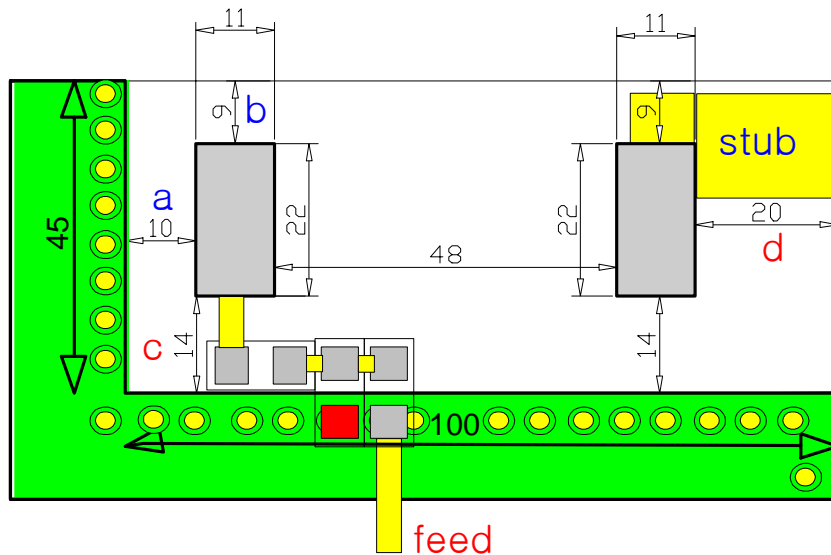


[Step 3 : Lumped shunt L = 1.8nH]

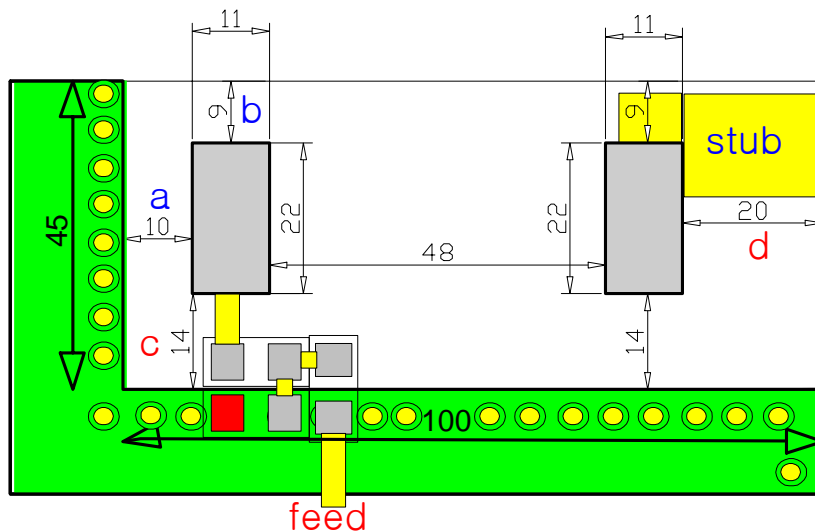
Lay-out Reference for 931 Antenna

- a, b : Minimum distance is required. It depends on customers' SMD procedure.
- c : Minimum 1mm is needed. If under 1 mm, the frequency bandwidth would be very narrow sharply.
- d : The distance under 1 mm is no problem in case the right side is open, but it requires over 1.5mm, if it is not open to the right side and there is a Ground.
- stub: We recommend to place in the most external side and to keep the lengths 1mm~3mm, and widths 1mm~2mm, which also depends on the PCB space to be adjusted.

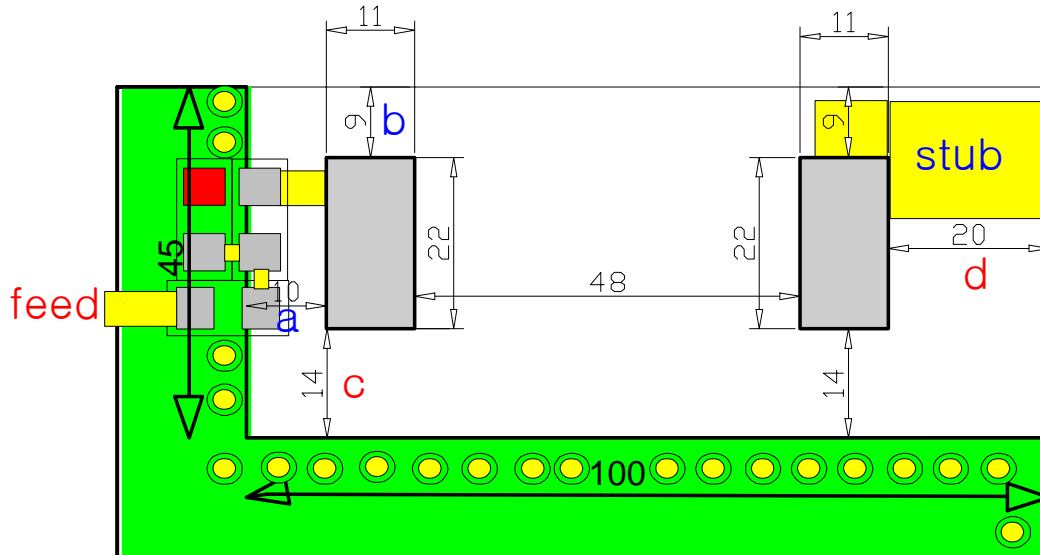
1) T type & bottom feed 1



2) T type & bottom feed 2



3) side feed 1



4) side feed 2

