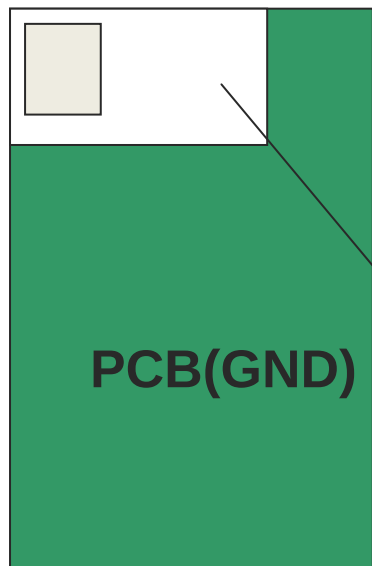


AH 086M555003

Antenna Design Guide



Wide Band Antenna



3.1 ~ 8.0GHz: AH 086M555003



Keep out area
(No GND Area)

Keep Out Area
(No GND Area)

Recommended Size
9.5 x 16.5 mm (157mm²) or more

Antenna Layout

Corner of PCB

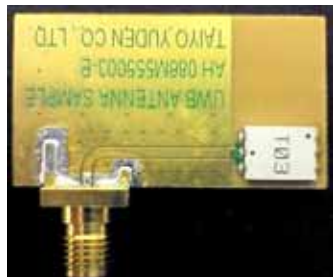
Radiation Pattern

Normal (Omni)

Data Sheet

■ Shapes

L= 8.0 ± 0.3 mm
W= 6.0 ± 0.3 mm
T= 1.0 ± 0.3 mm



*on Taiyo Yuden's Evaluation Board
(36 x 20 mm)

*Element-GND Distance : 1.5 mm

■ Feature

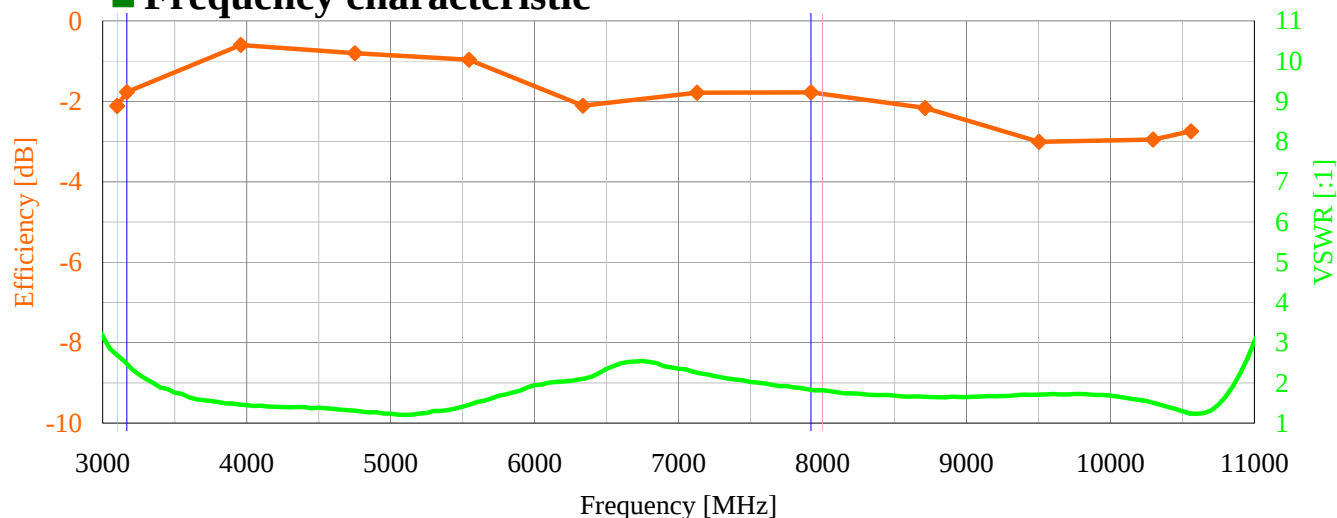
- * Small
- * Wide Band
- * High Performance



Actual data

Efficiency : -0.6 ~ -2.1dB (87 ~ 62%)
Peak Gain : 2.6 ~ 0.7dBi
@3168~7920MHz

■ Frequency characteristic



■ Electrical Characteristics

				3100MHz	3168MHz	3960MHz	4752MHz	5544MHz	6336MHz	7128MHz	7920MHz	8712MHz	9504MHz	10296MHz	10560MHz
Efficiency [dB]				-2.1 (61%)	-1.8 (66%)	-0.6 (87%)	-0.8 (83%)	-1.0 (80%)	-2.1 (62%)	-1.8 (66%)	-1.8 (66%)	-2.2 (61%)	-3.0 (50%)	-2.9 (51%)	-2.7 (53%)
Peak gain [dBi]				1.8	2.0	2.6	2.1	1.5	0.8	1.0	0.7	0.4	-0.8	-0.1	0.5
Average gain [dBi]	XY-plane	TX-H		-4.1	-4.0	-2.1	-2.5	-4.3	-6.1	-7.0	-6.1	-5.6	-4.9	-4.7	-4.1
		TX-V		-5.5	-5.7	-4.3	-3.5	-2.7	-3.9	-3.3	-2.6	-3.3	-7.7	-11.7	-12.4
	YZ-plane	TX-H		-3.8	-3.4	-2.7	-2.6	-2.3	-3.4	-3.2	-3.5	-4.1	-5.0	-4.3	-4.3
		TX-V		-14.3	-14.8	-20.1	-19.7	-14.1	-13.9	-11.8	-13.0	-13.4	-14.8	-17.6	-17.0
	ZX-plane	TX-H		-9.8	-9.5	-8.3	-6.5	-5.3	-7.5	-7.7	-6.8	-6.8	-9.7	-14.0	-13.8
		TX-V		-1.1	-0.7	0.5	-0.9	-2.3	-3.5	-2.6	-3.6	-3.6	-3.2	-2.0	-1.6

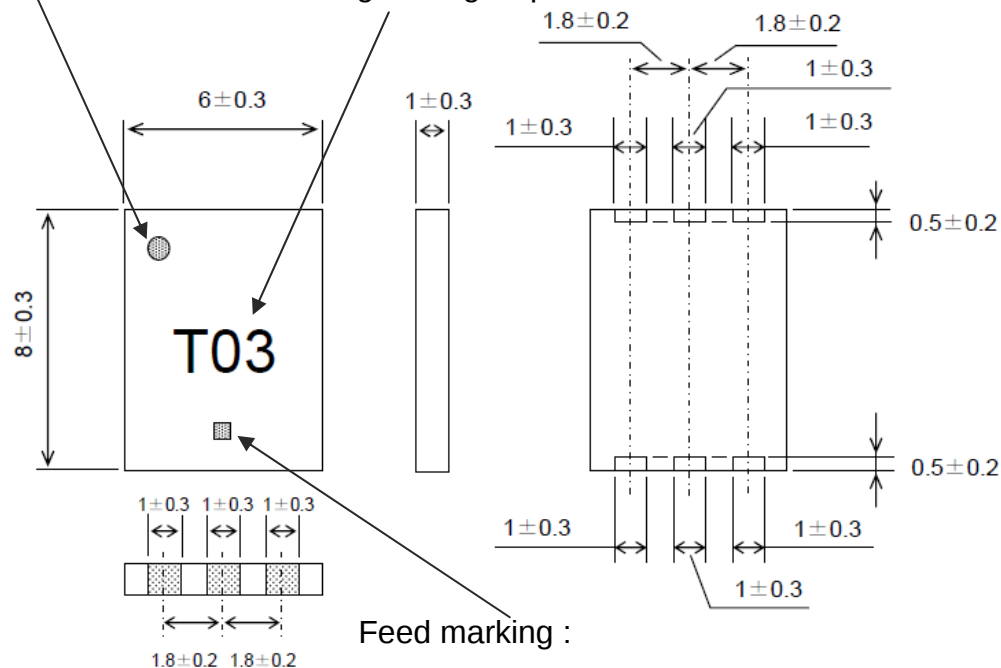
Dimensions and Land pattern

External Dimensions

Direction marking : 0.6mm ϕ

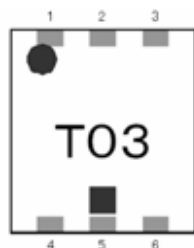
An intestinal pattern code:

1.6mm height 3-digit alpha numeric



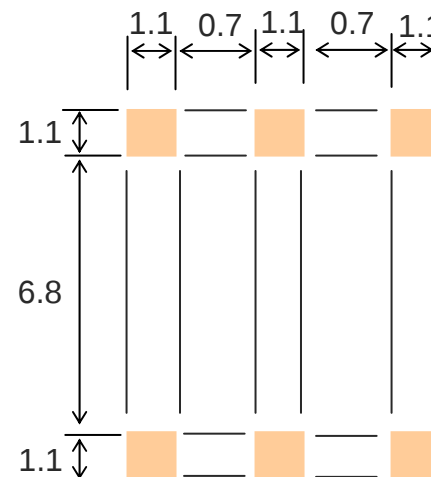
Feed marking :

Input terminal is prescribed 0.5mm $\square$



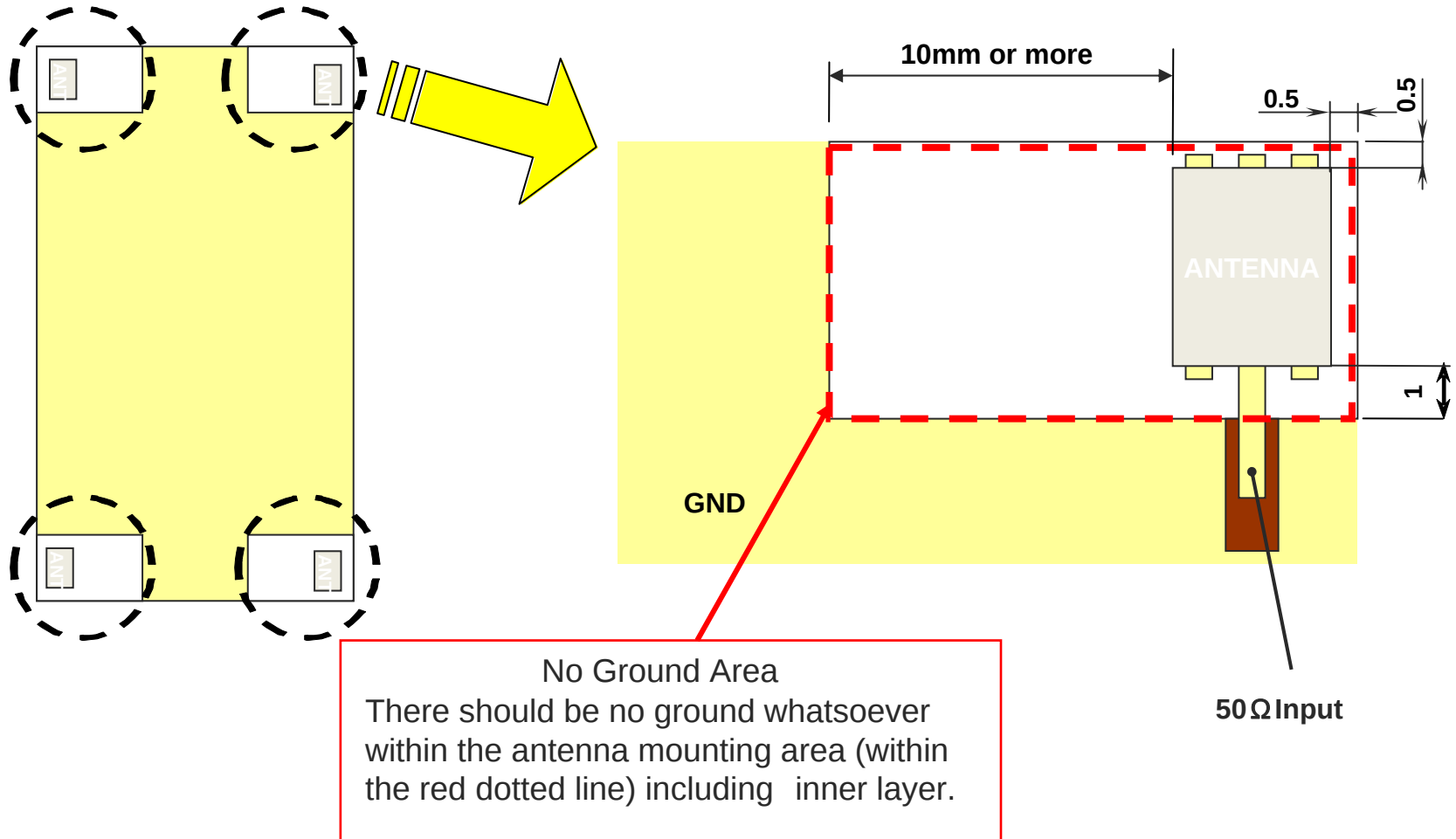
1	NC	4	NC
2	NC	5	FEED
3	NC	6	NC

Recommended Land Pattern



Design Guide

Recommended Pattern Layout



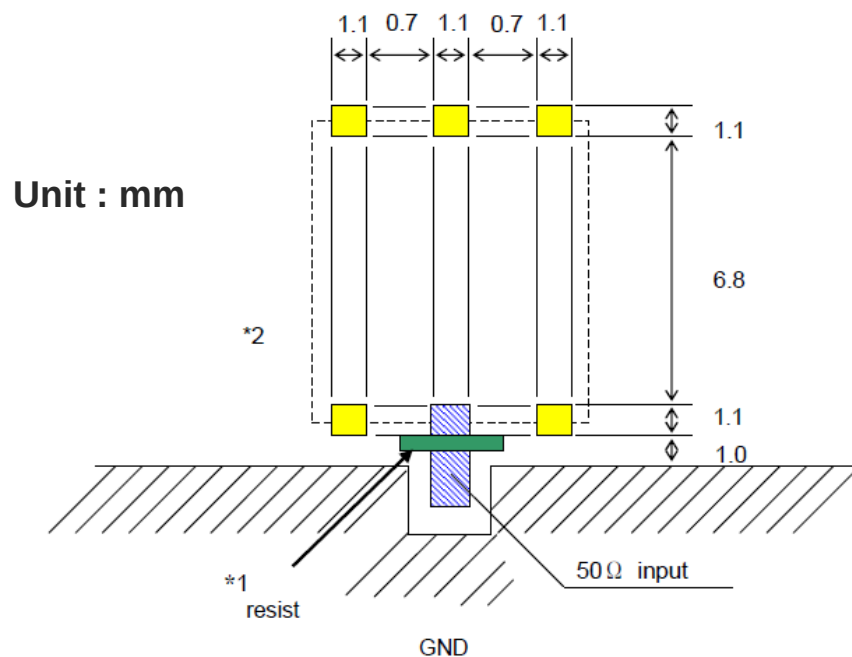
Our recommendation is mounting position had better on the corner of PCB.

Design Guide

Precautions

- . Surface GND layer around the antenna area should be connected with inner GND layer via through hole.
- . Thickness of PCB can be flexible.

Recommended land pattern



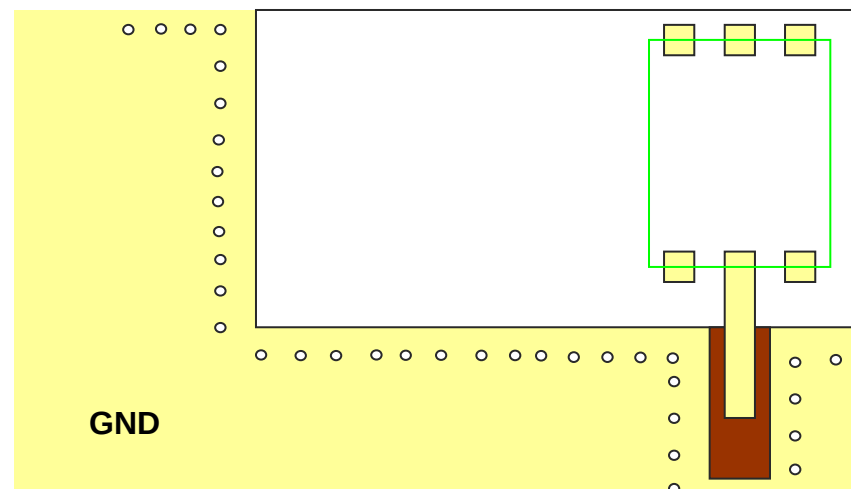
Land for input terminal

Land for NC terminal

*1: Soldering area shall be ensured by the solder resist.

*2: Do not arrange the surface pattern and internal layer pattern on antenna mounting area and its surrounding area.

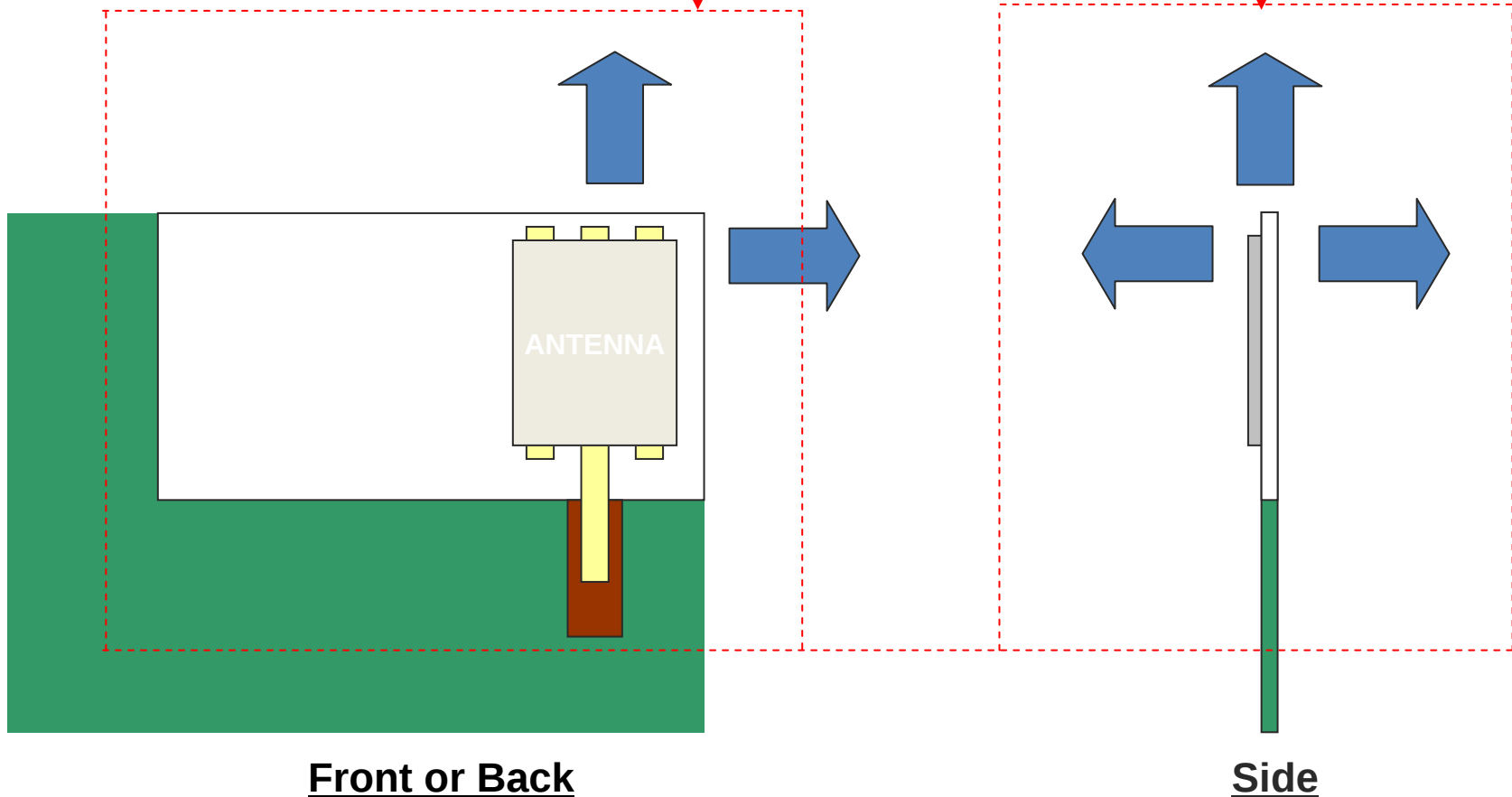
Example of through hole



Design Guide

Metal Avoidance Area

Please do not set close to a metal housing, paint including the metal, board GND, the metal chassis, etc.



TAIYO YUDEN