Application Note no.003

Chip Antenna Series

GPS Chip Antenna

ACD-5036-A3-CC-S

Prepared	Checked	Approved
Leeting	**David Tsai**	**David Tsai**

INPAQ Technology Co., LTD
http://www.inpaq.com.tw
TEL :+886-37-585-555
FAX : +886-37-585-511

INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

Application Note

GPS Chip Antenna – ACD-5036-A3-CC-S

Revision History: 2009-07-02 Rev.A1

Previous Version :

Page	Subjects (major changes since last revision)	Version
All	Make up all document	A0
2	Revise recommended PCB layout	A1

INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

ACD-5036-A3-CC-S Application Note

Catalog

INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

Applications

This antenna is designed for GPS application and it's suitable for cellular phones, PDA, notebook, navigator, and all devices which have GPS function.

Features

- Omni-directional radiation
- High Efficiency
- Low profile and compact size(5.2 x 3.7 x 0.7mm)
- Low cost
- Lead free soldering compatible
- RoHS compliant
- Tape and reel packing

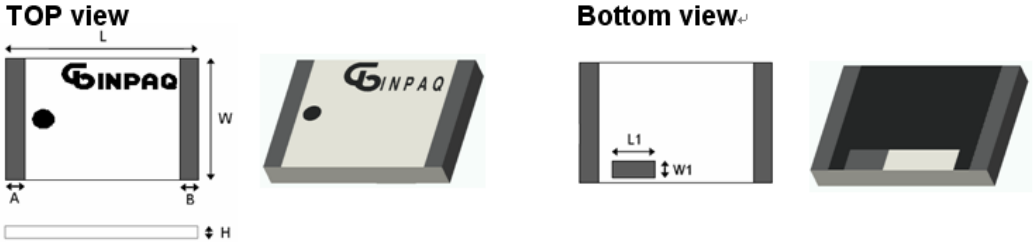
Electrical Characteristics

Electrical Specification*	
Center Frequency	1575.42MHz
Frequency Range	1550MHz~1600MHz (S11≦ -10dB)*
Polarization	Linear
Pattern	Omni-Directional
Ref. Impedance	50 ohm
Peak Gain	3.40 dBi (typ.)@1575.42MHz
Efficiency [%]/[dB]	83.1% / 0.13 dBi (typ.)@1575.42MHz
Size	5.2mm x 3.7mm x 0.7mm

* Electrical characteristic depends on INPAQ evaluation board with matching circuit.

* The dimension of evaluation board is 80 x 40mm with 15x6.3mm clearance area.

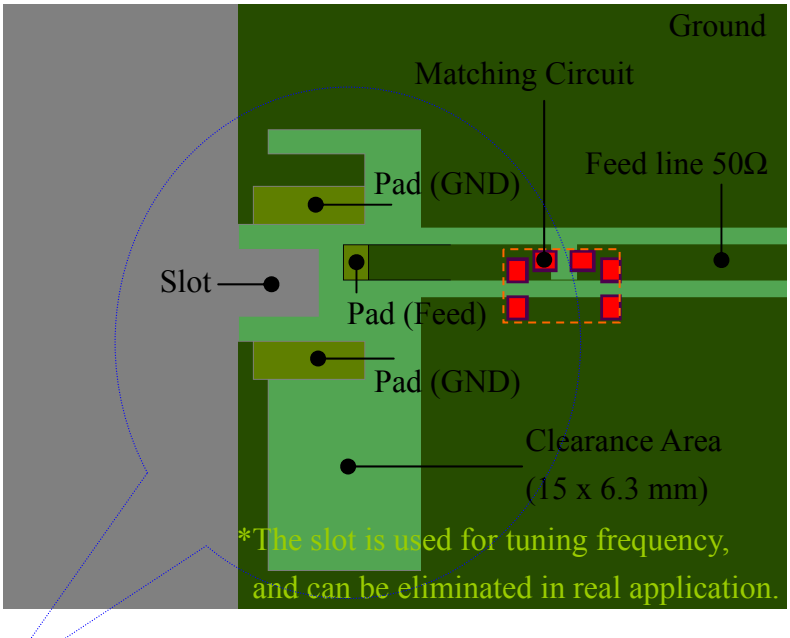
Antenna Dimension (unit:mm)

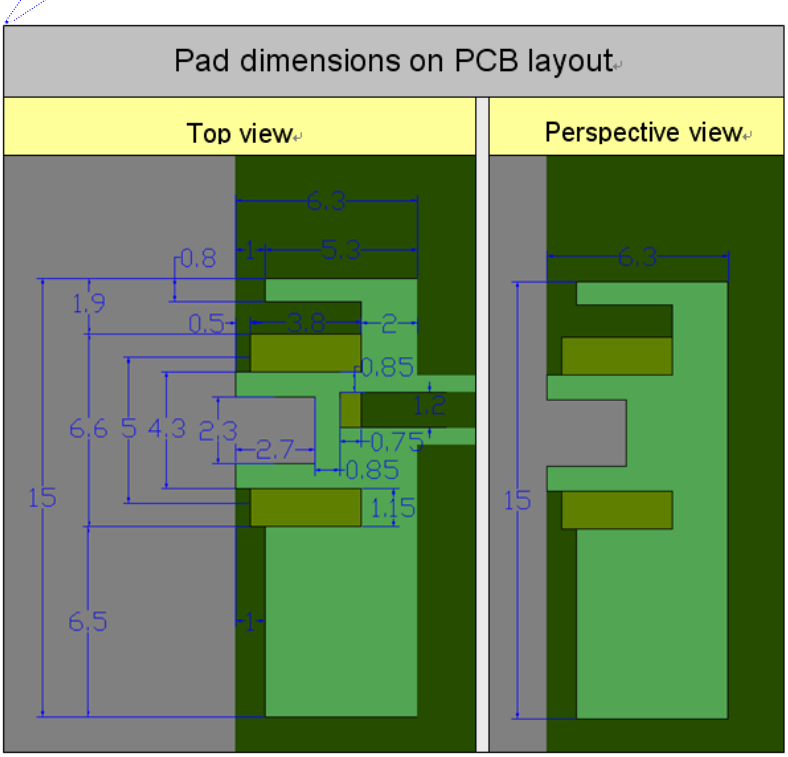


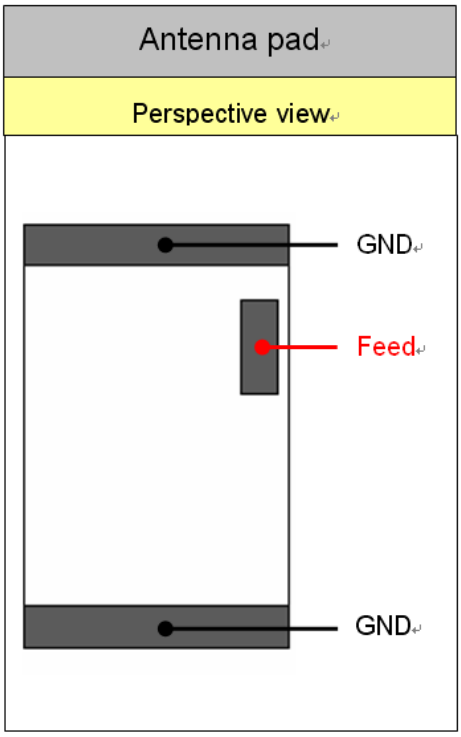
Chip Antenna	L	W	A	B	L1	W1	H
ACD5036A3	5.2±0.3	3.7±0.3	0.45±0.25	0.45±0.25	1.1±0.20	0.55±0.20	0.70±0.15

INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

Recommended PCB layout(unit:mm)







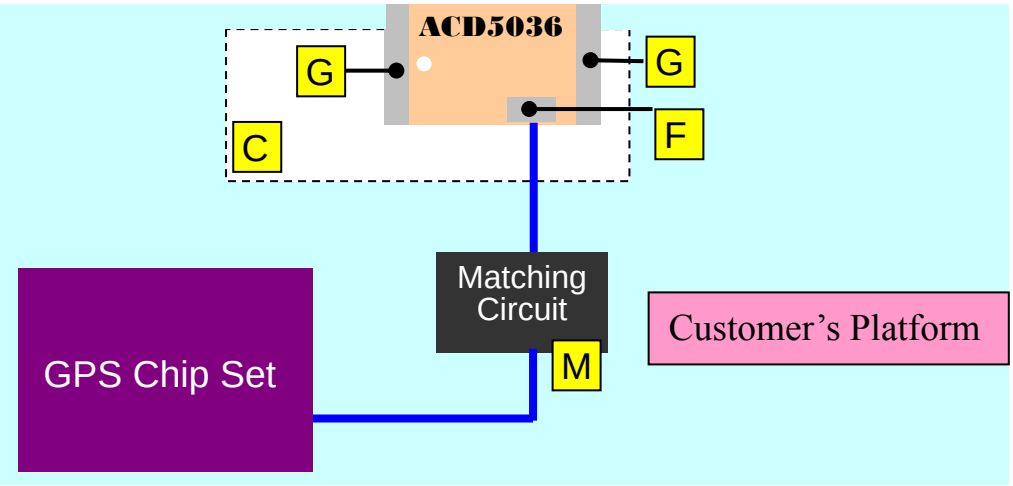
PCB pad

Terminal name	Terminal Dimensions
Pad (Feed)	1.2 X 0.75
Pad (GND)	3.8 X 1.15
Pad (GND)	3.8 X 1.15

Antenna pad

Terminal name	Terminal Dimensions
Feed	1.1 X 0.55
GND	3.7 X 0.45
GND	3.7 X 0.45

Layout Description



F. Feeding Pad

The signal from system must feed into the feeding pad.

G. Ground Pad

This pad must connect to ground plane of PCB.

C. Clearance Area

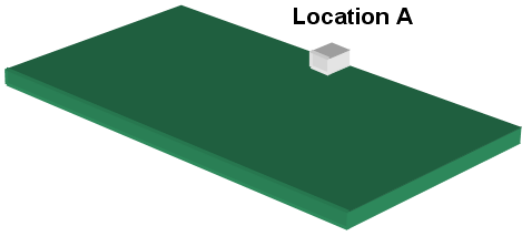
To achieve antenna performance, the clearance area is necessary and all metallization should be removed from all PCB layers.

M. Matching Circuit

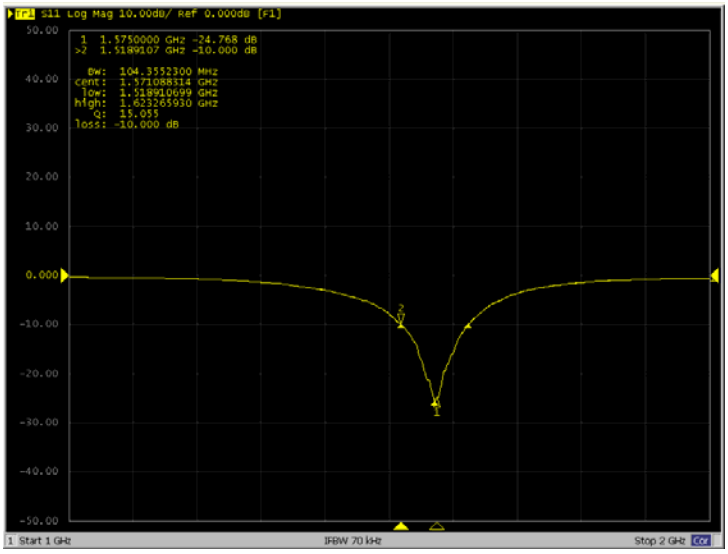
Please keep the pads for PI-matching circuit to reduce return loss and shift the band to meet GPS application.

INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

Performance on Middle of Long Side

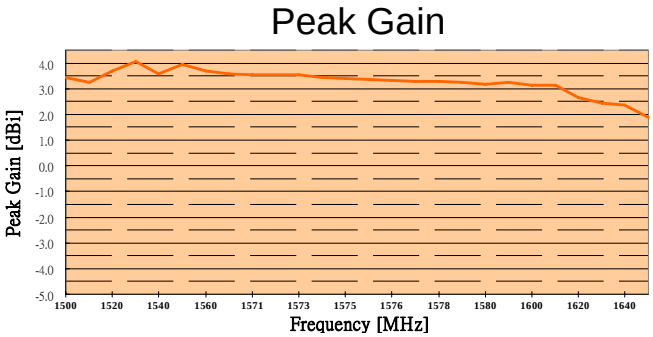


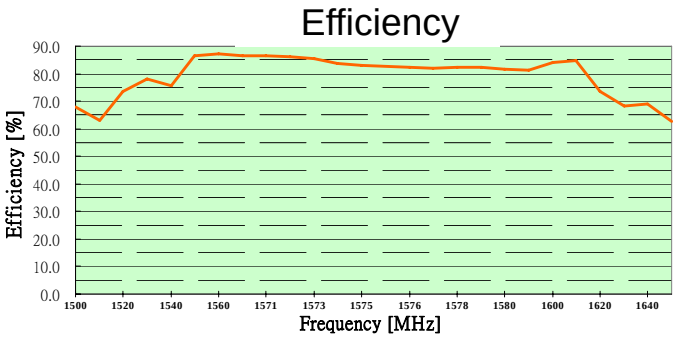
Typical S11



Frequency	S11
1575 MHz	-24.77
1518 MHz	-10.00
1623 MHz	-10.00

Typical Free Space Peak Gain and Efficiency





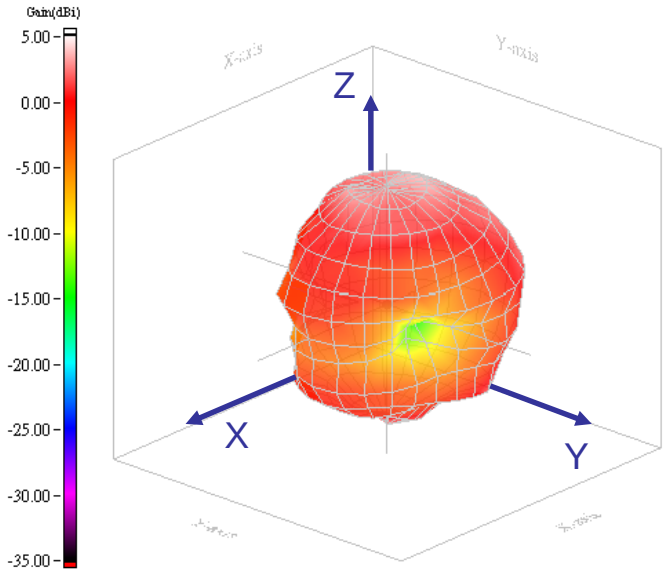
Frequency	Peak Gain	Efficiency
1570 MHz	3.57 dBi	86.56
1575 MHz	3.40 dBi	83.10
1580 MHz	3.19 dBi	81.49

INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

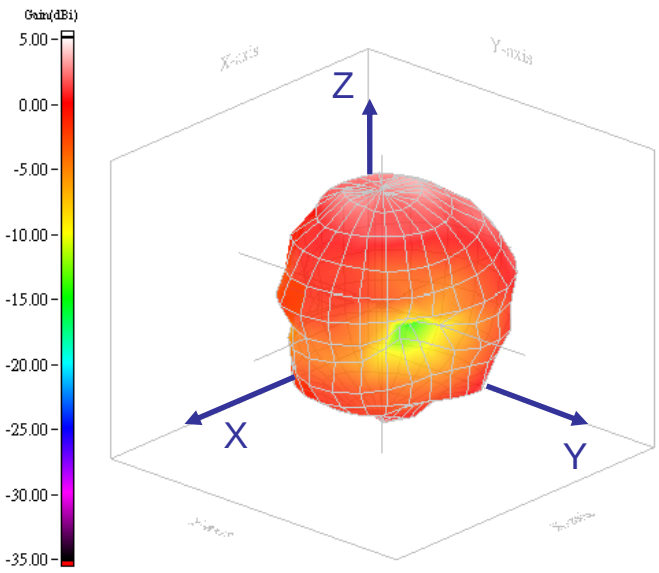
Typical Free Space Radiation Pattern

3D Radiation Pattern

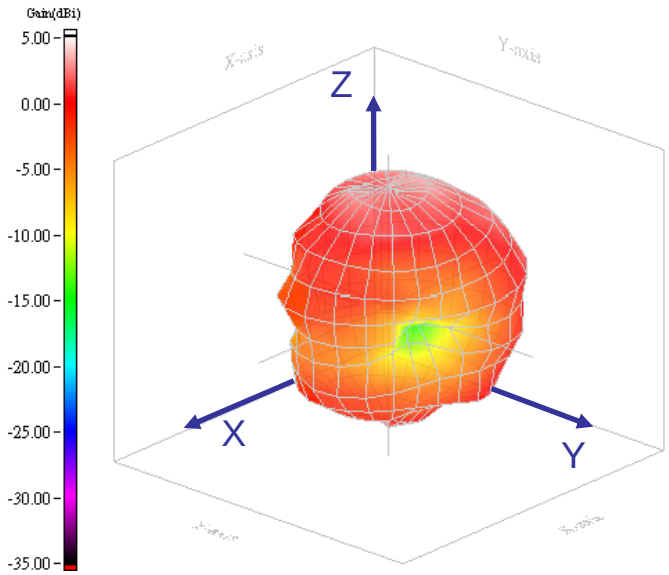
1570 MHz

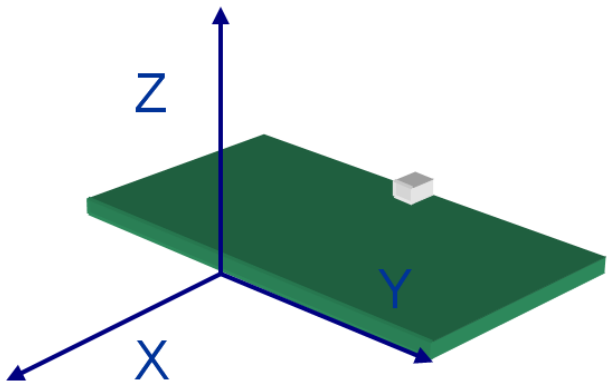


1575 MHz



1580 MHz



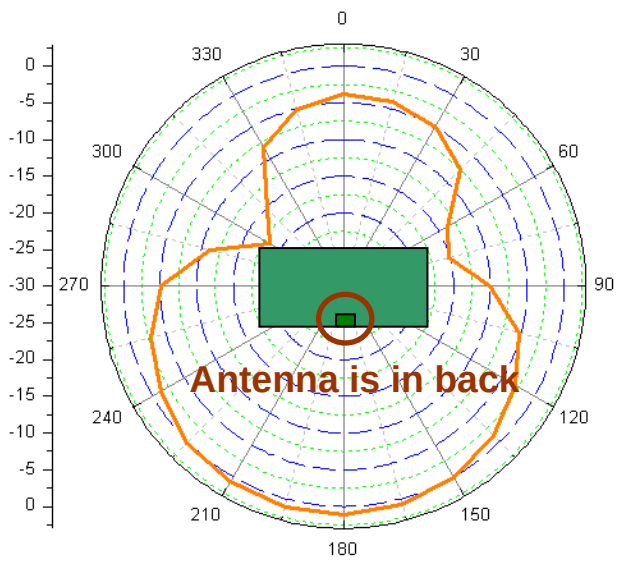


INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

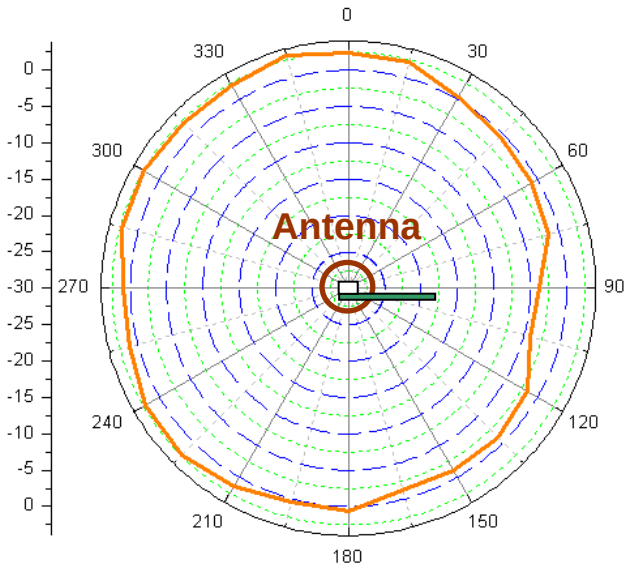
Typical Free Space Radiation Pattern

2D Radiation Pattern(1575 MHz)

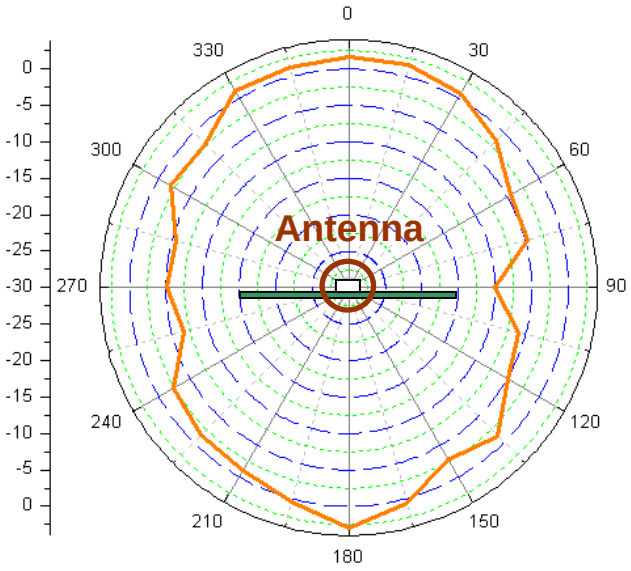
X-Y Plane

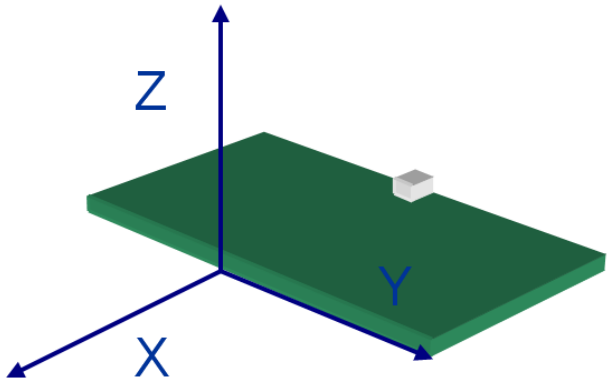


X-Z Plane



Y-Z Plane

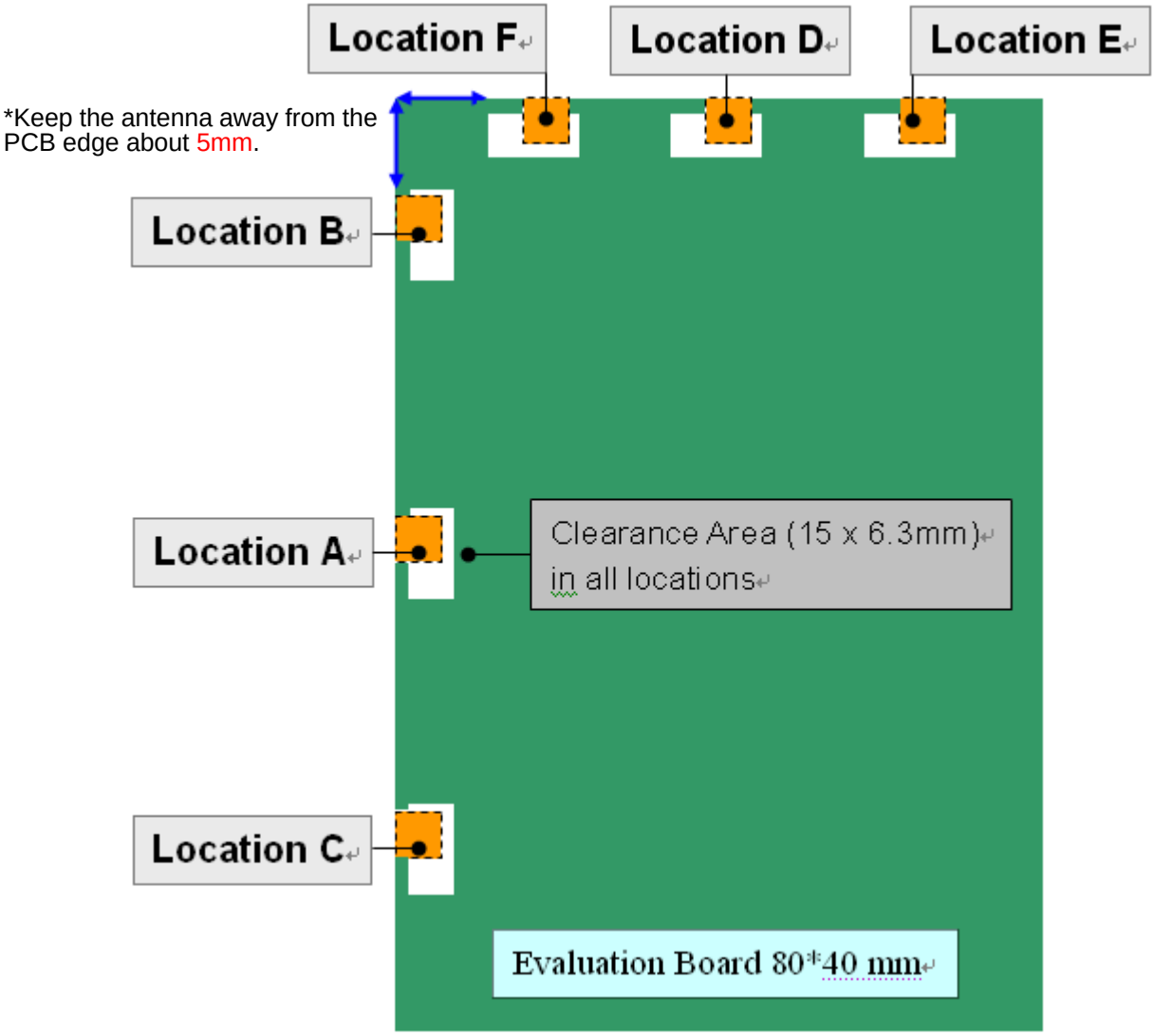




INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

The Efficiency and Bandwidth for Different Location

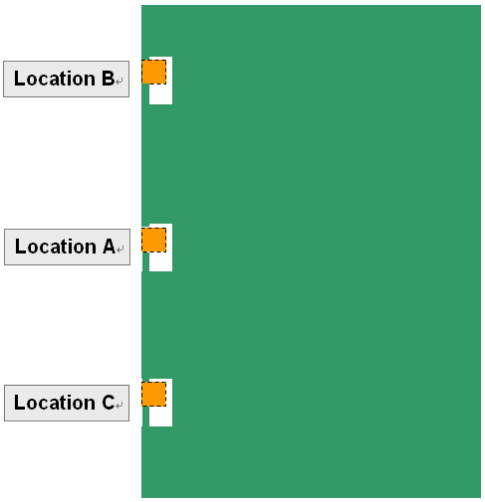
* All electrical characteristic depend on INPAQ 80 x 40mm evaluation board with matching circuit.

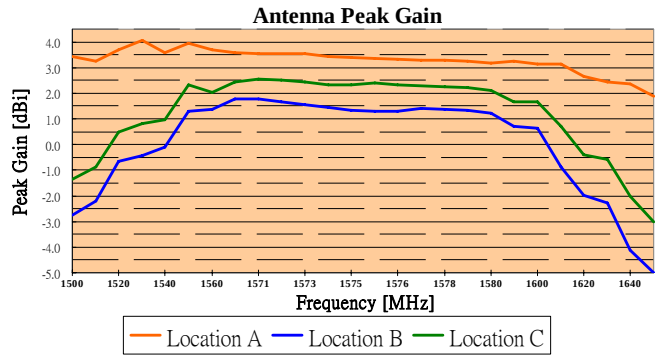


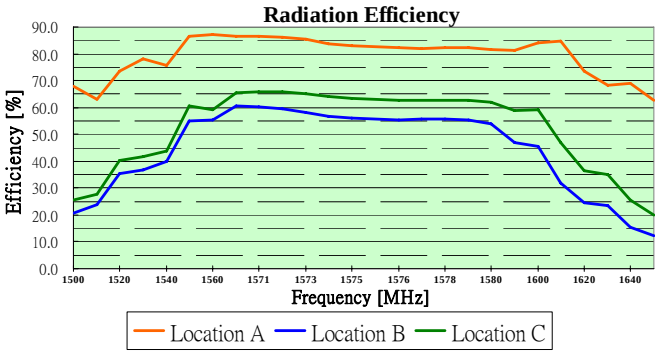
Locations / Test Item			Long side			Short side		
			A	B	C	D	E	F
Bandwidth [MHz] S11<-10			104	33	42	23	22	20
Gain	Linear [dBi]	Peak	3.40	1.35	2.32	0.08	0.35	-0.46
		Avg.	0.13	-2.08	-1.31	-2.57	-2.75	-2.74
Efficiency	Linear [%]		83.1	55.87	63.31	55.99	53.75	53.94

INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

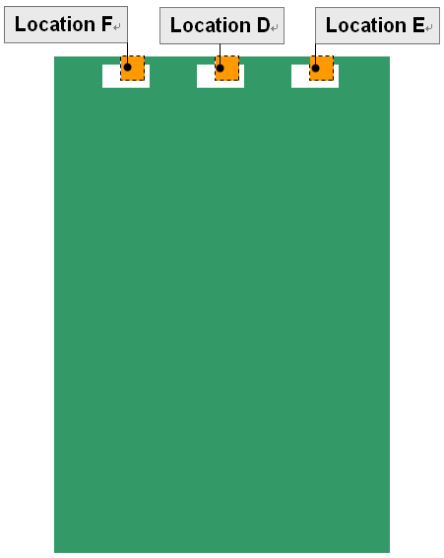
Peak Gain and Efficiency on Long Side

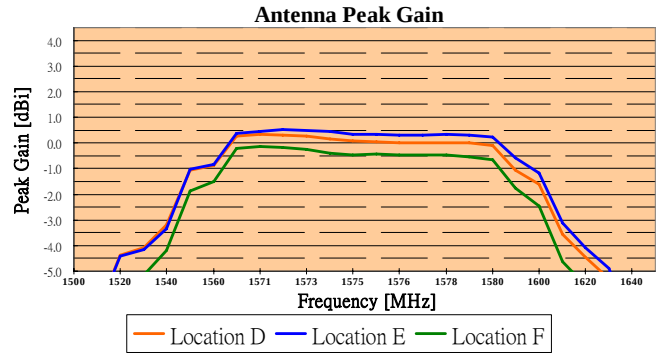


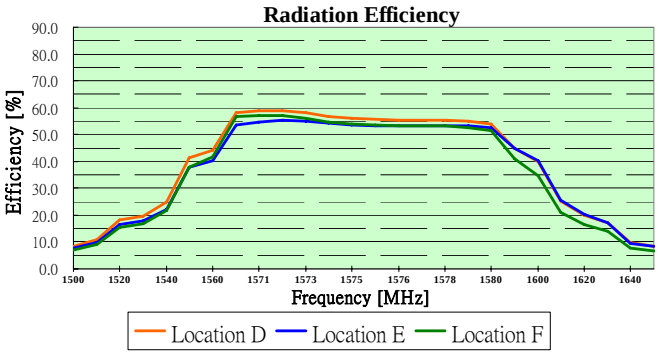




Peak Gain and Efficiency on Short Side



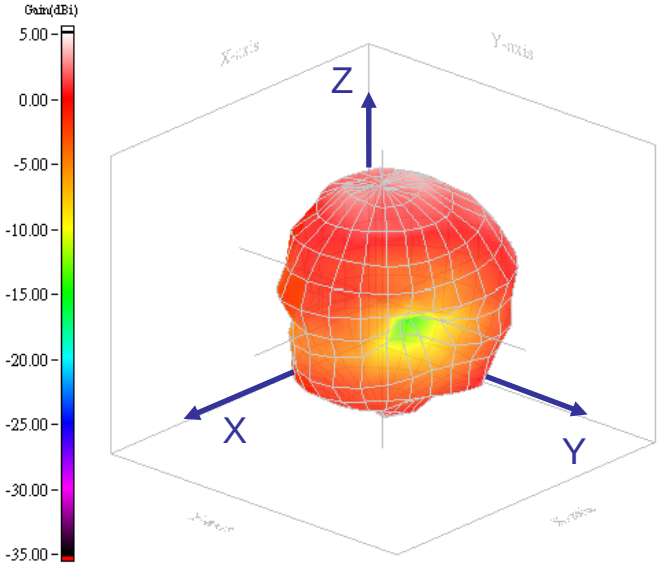




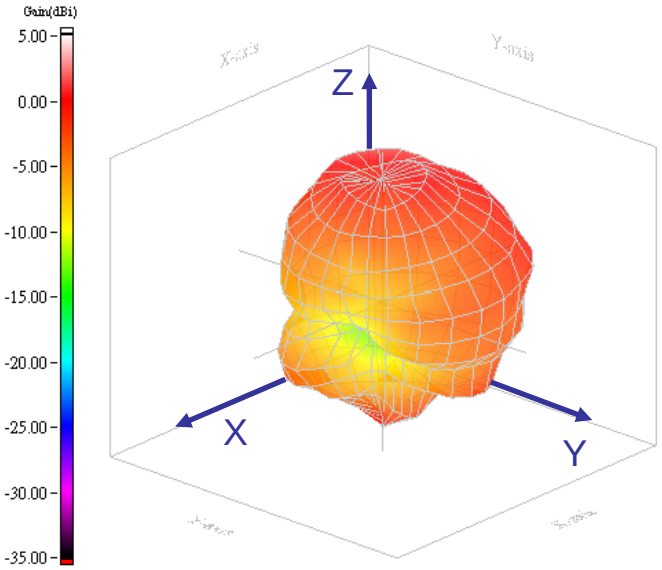
INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

3D Gain Pattern on Long Side(1575 MHz)

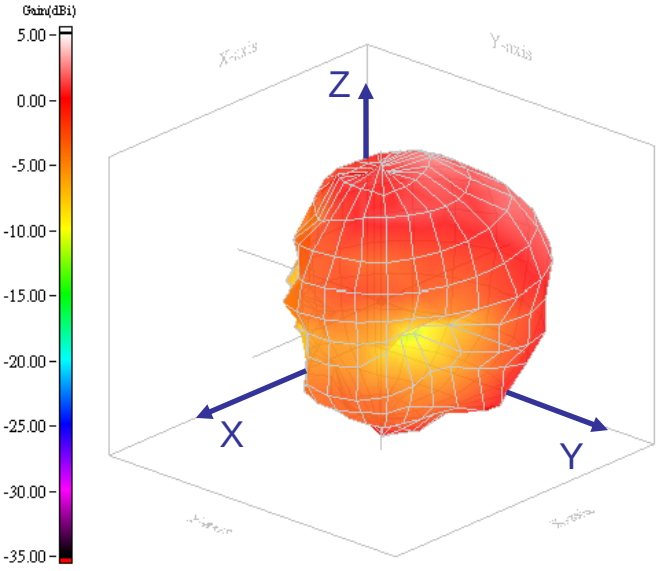
Location A

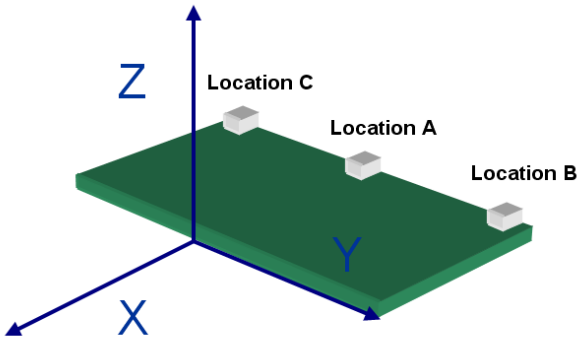


Location B



Location C

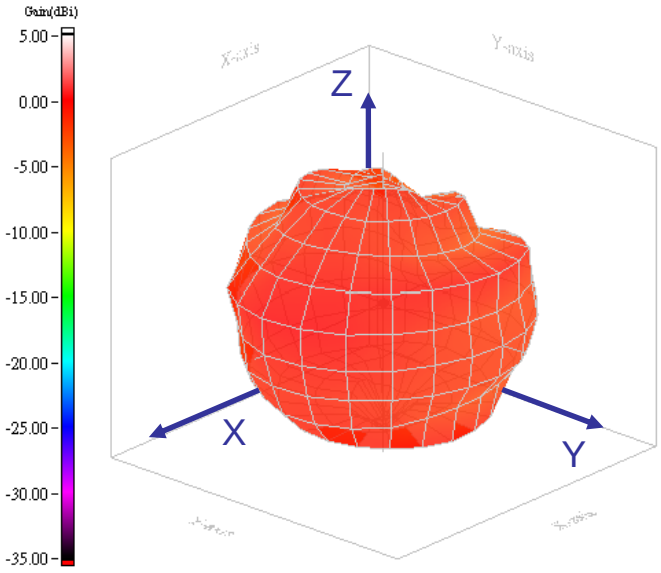




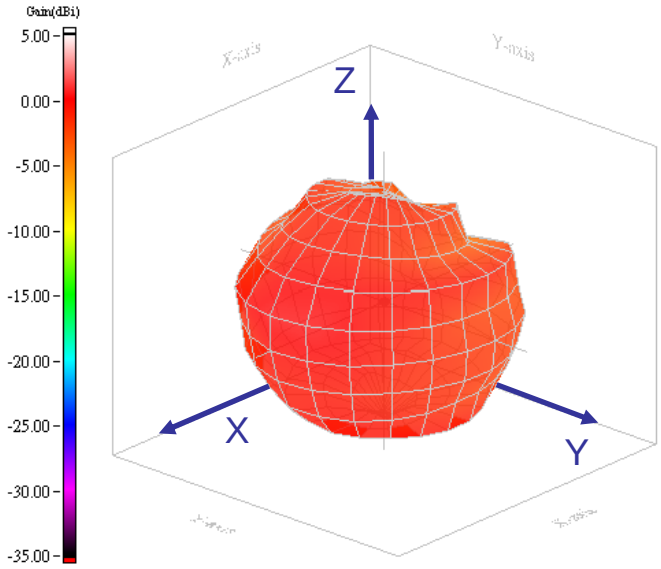
INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

3D Gain Pattern on Short Side

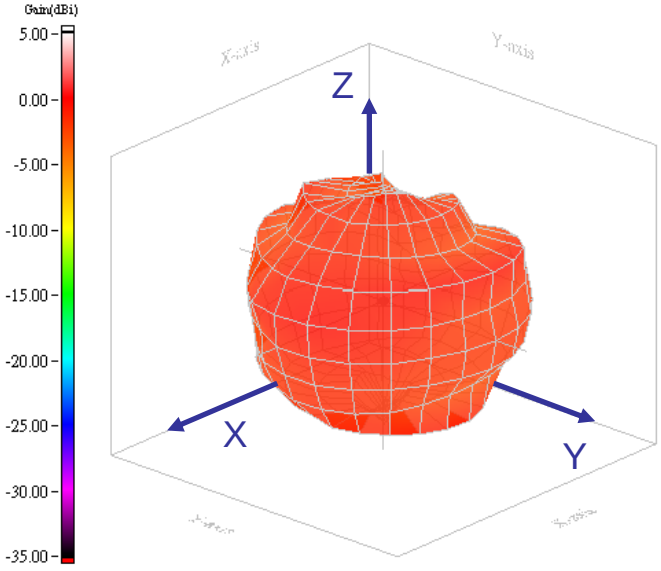
Location D

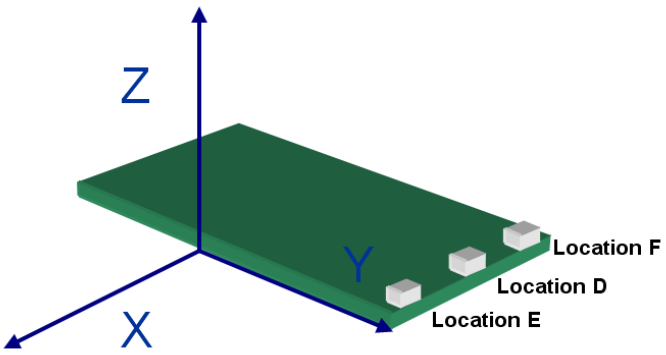


Location E



Location F

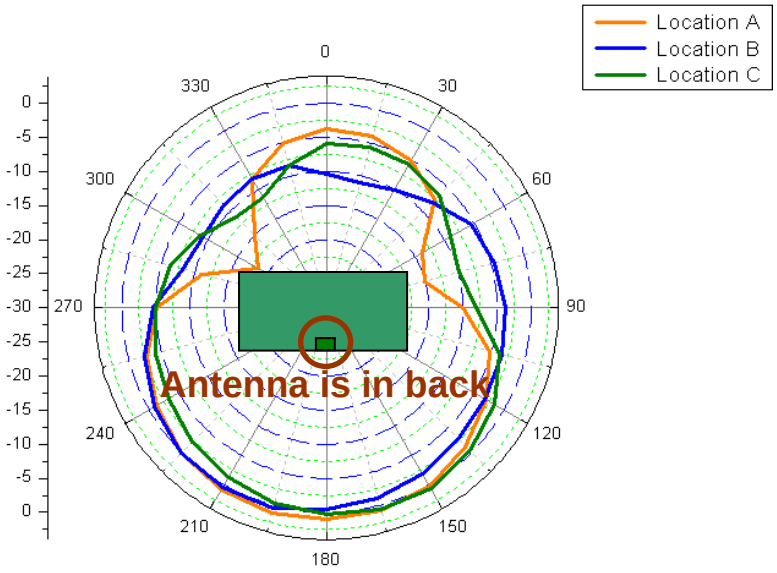




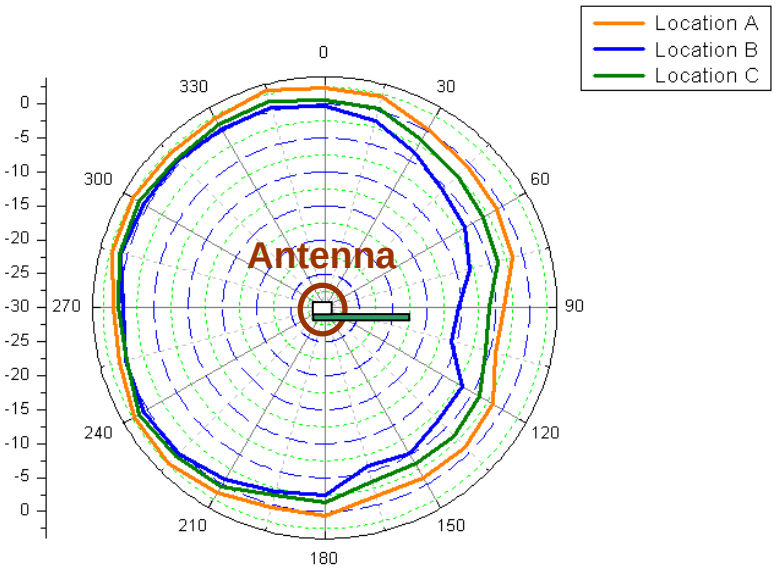
INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

2D Gain Pattern on Long Side(1575 MHz)

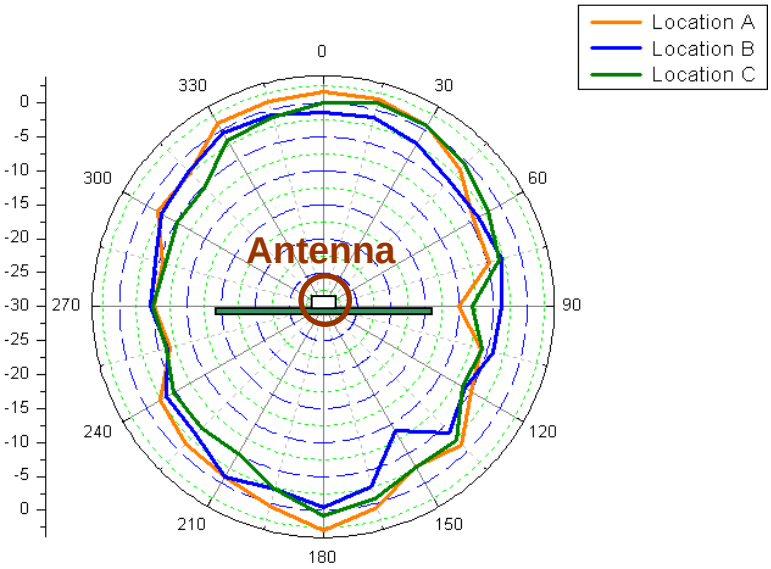
X-Y plane

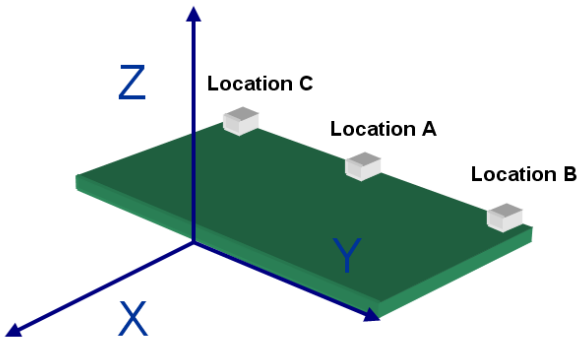


X-Z plane



Y-Z plane

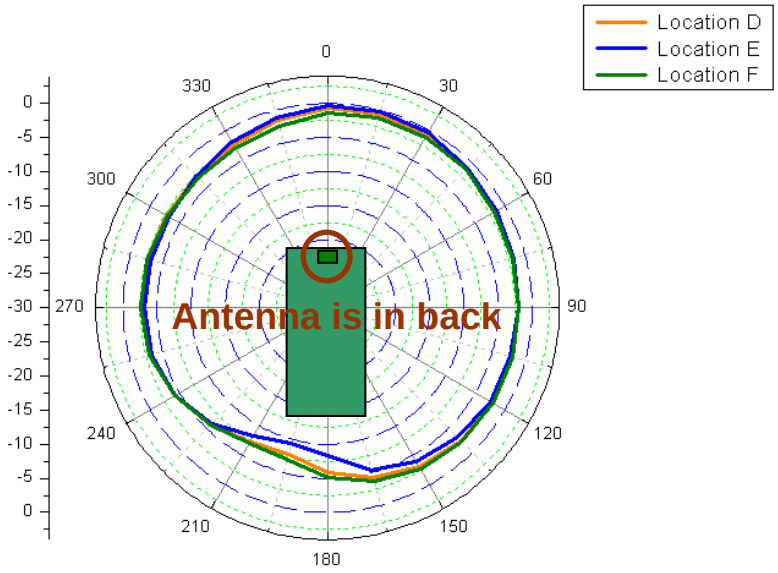




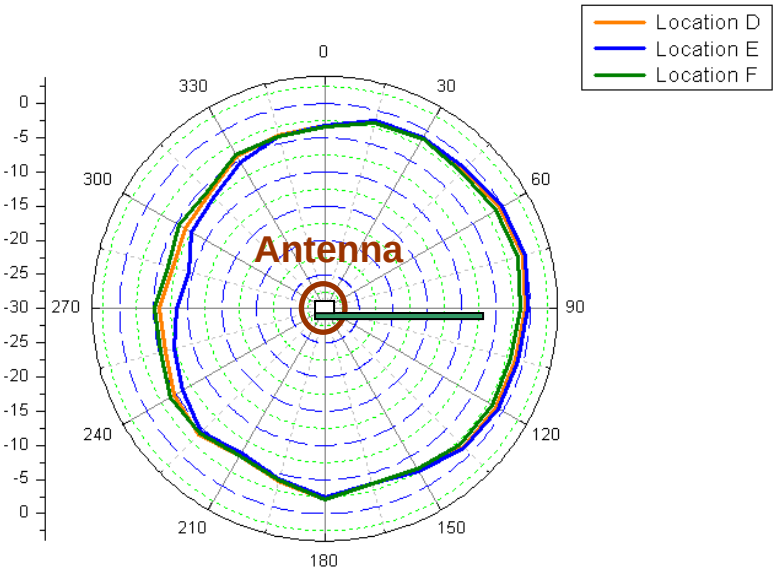
INPAQ Taiwan
TEL:+886-37-585-555
FAX: +886-37-585-511

2D Gain Pattern on Short Side(1575 MHz)

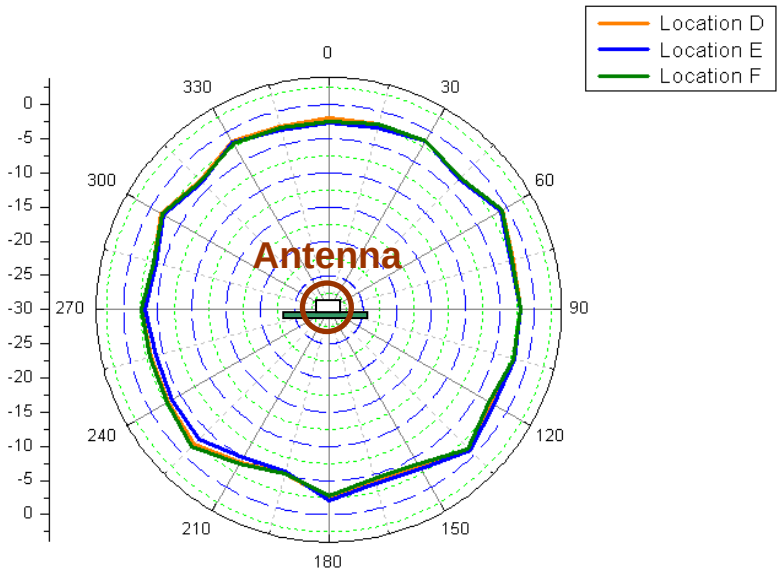
X-Y plane

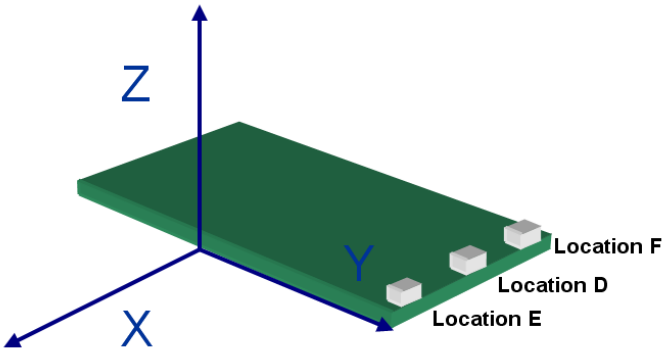


X-Z plane



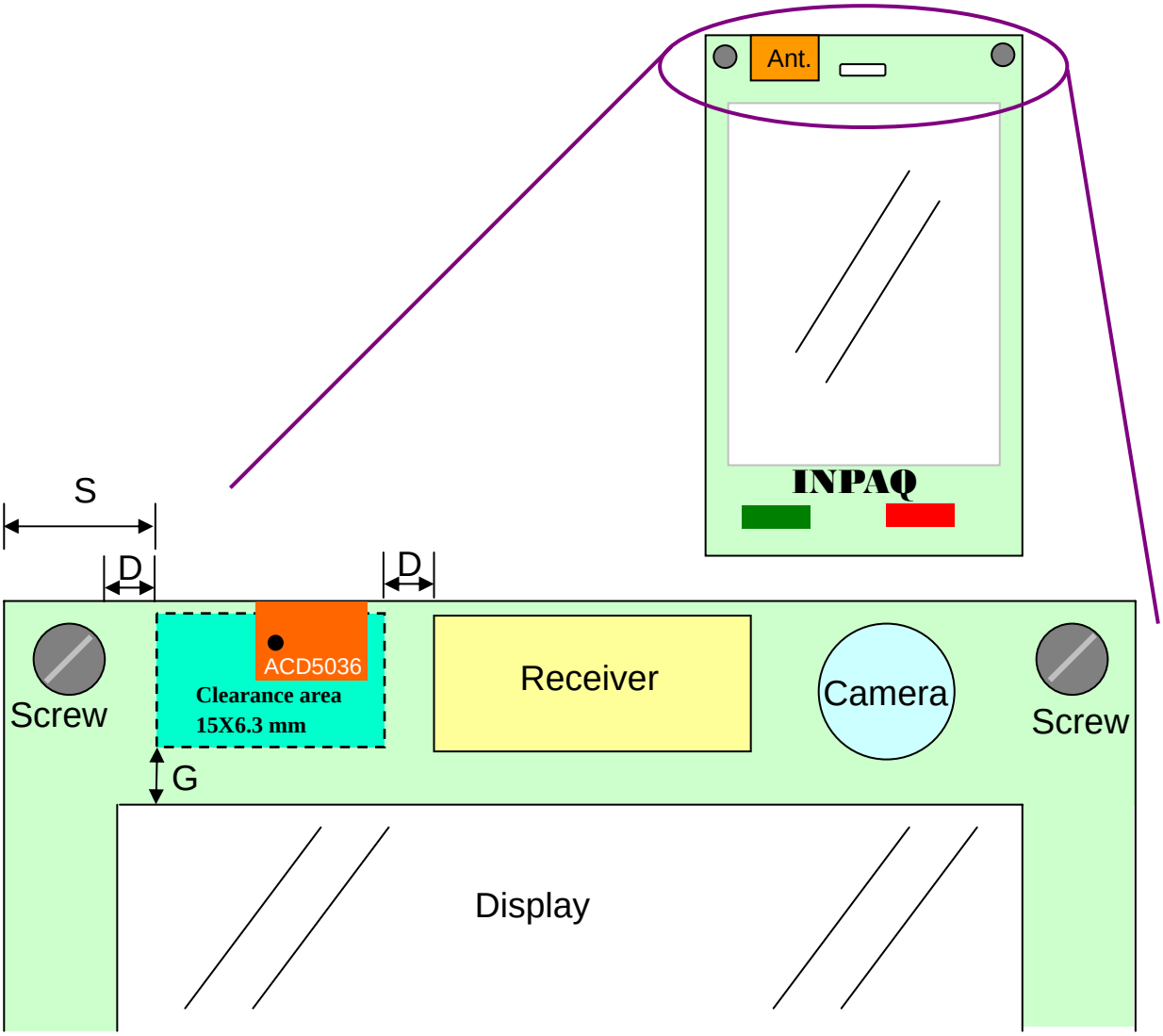
Y-Z plane





Mobile Phone Applications

- For the mobile phone applications, because most of the key components are arranged along the long side of the PCB, there is no space to place our antenna. So we move the antenna to top edge of PCB as showed as follow picture. And the impedance of PCB in top edge is smaller than in long side edge, we get narrower bandwidth and lower performance than in long side of PCB. But we still get a workable performance by arranging antenna and components in a reasonable position.



Symbol	Suggested Distance	Remark
S	≧5mm	The distance between PCB edge and antenna edge
D	≧1mm	The distance between antenna and receiver(or shielding case)edge
G	≧1mm	The edge of display must keep away 1mm from antenna edge.

PND Applications

- For the PND or Navigator applications, GPS antenna usually place at the long side of PCB. To make the device thinner, it usually cut a part of PCB to put the battery in it and result in a shorter PCB than mobile phone application. In order to increase the performance of GPS antenna, we suggest to keep some part of PCB to make it look as L-shape as figure E. However, the distance between panel and PCB will affect the antenna performance, we suggest keep the panel away from antenna edge at least 3mm in distance as Figure F.

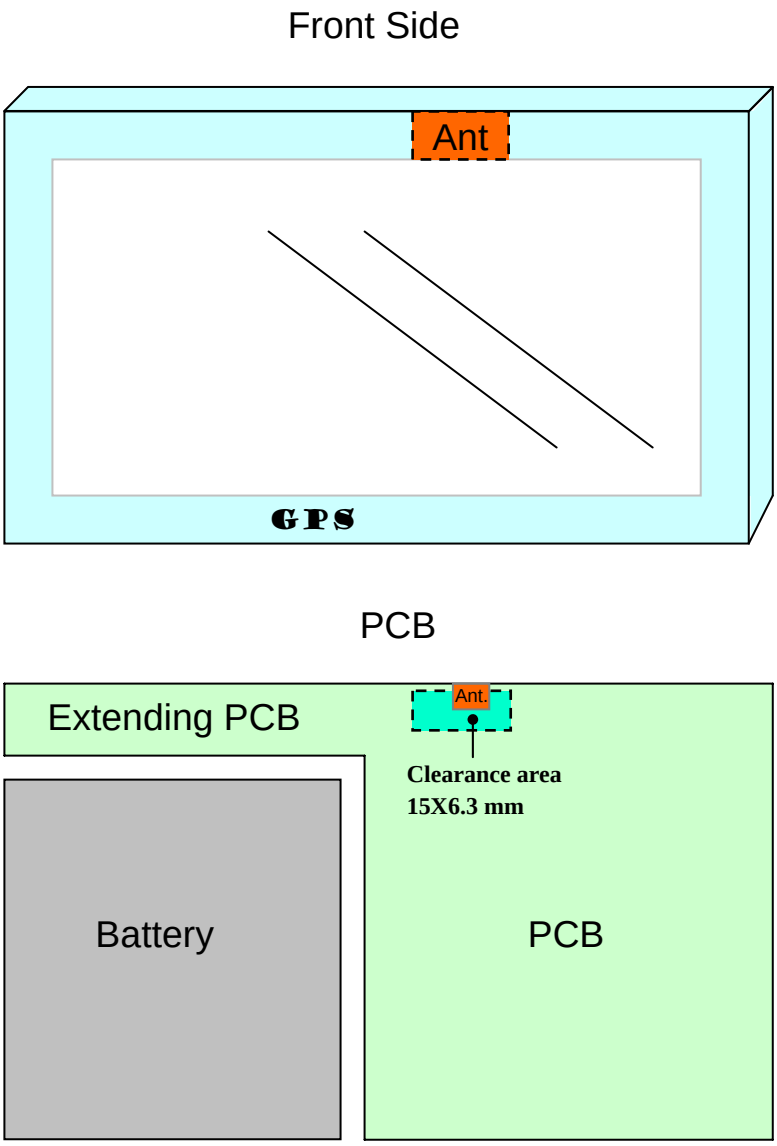


Figure E. Make the extending PCB to get the better performance

Back Side

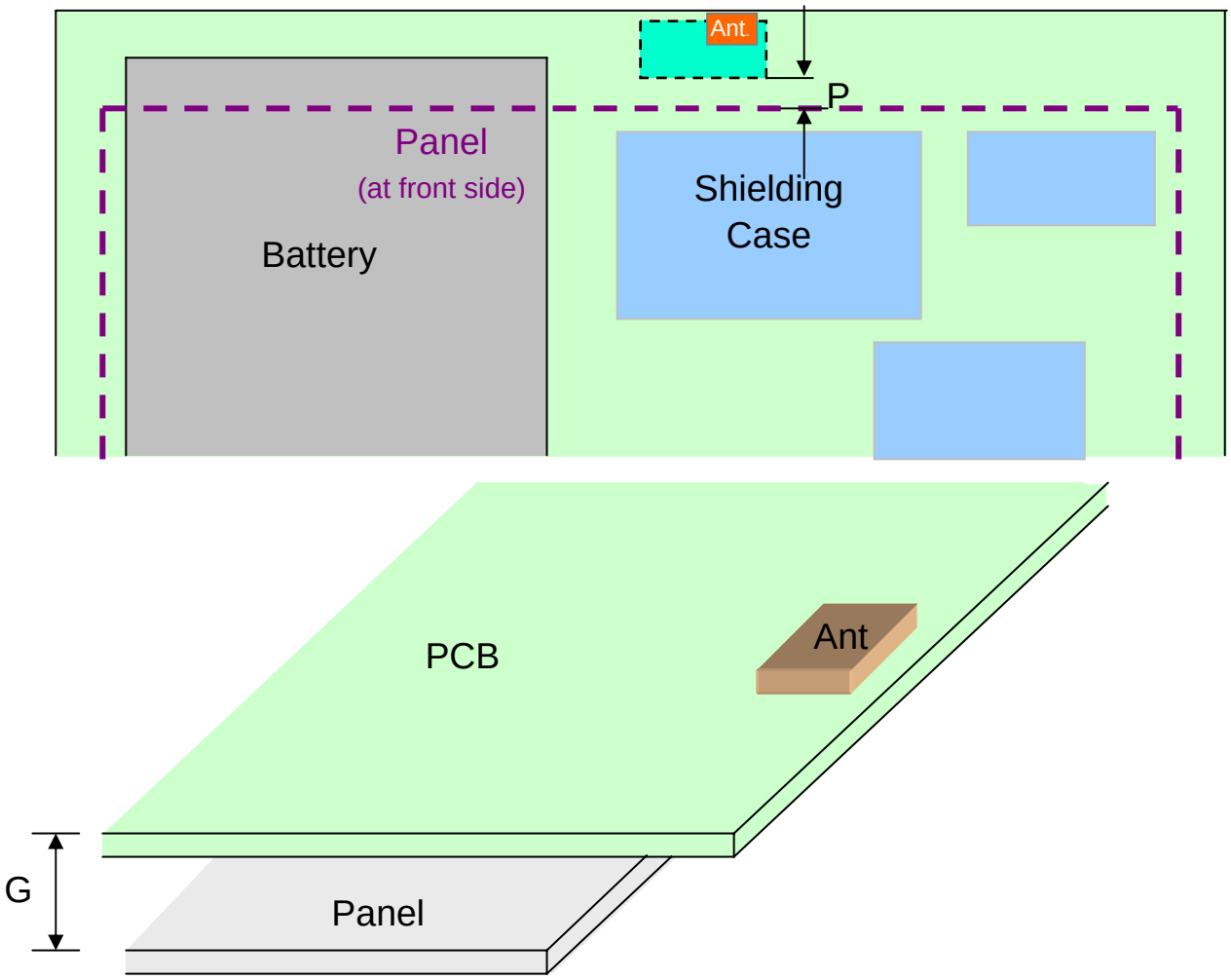
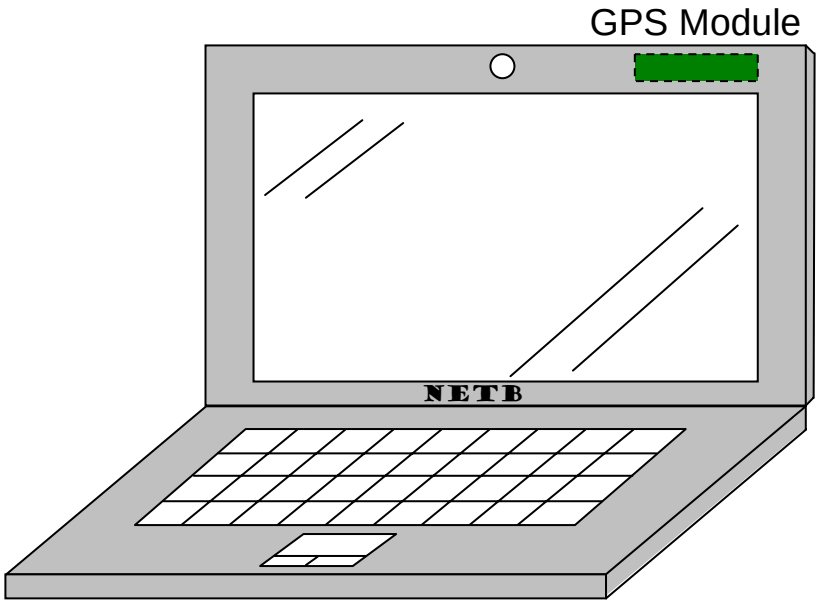


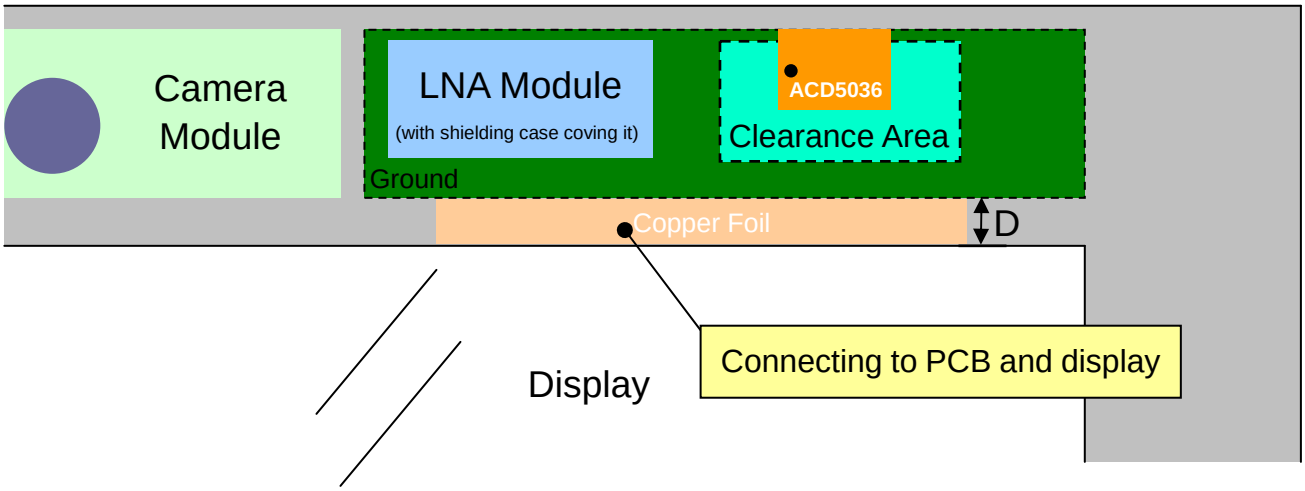
Figure F. Keep the panel away from the PCB more than 3mm.

Symbol	Suggested Distance	Remark
P	≧1mm	The edge of display must keep away 1mm from antenna edge.
G	≧3mm	The distance between antenna and panel metal

Notebook Applications

- For the notebook applications, the space is too small to place a larger PCB. As we know, the smaller PCB we have, the worst antenna performance we get. But according to characteristic of this antenna, because the radiation efficiency depends on the size of the metal layer, so we can extend metal layer from PCB to panel by using copper foil. And if the radiation plane can be extended to metal of panel, the PCB size becomes a minor factor of antenna performance; it means we can use smaller PCB to get the similar performance. By the way, the cable which connects from PCB to main board must fix along the edge of display, and shorter cable will get the better performance due to its cable loss.





Symbol	Suggested Distance	Remark
D	≧1mm	The distance between GPS Module and the edge of panel.

Contact Information

INPAQ Technology Co, LTD

Taiwan Chunan

ADD: No.11, Ke-Yi St., Chunan, Miaoli 35059, Taiwan (R.O.C.)
TEL : +886-37-585-555
FAX : +886-37-585-511
E-Mail : info@inpaq.com.tw
Web site : http://www.inpaq.com.tw

USA Office

ADD: 21 Echo Brook Road, Rochester, NH 03839, U.S.A.
TEL:+1-603-332-6222
FAX:+1-603-509-2900

Korea Office

ADD: 221 Raemian Seocho Univill, 1445-4, Secho-Dong, Secho_gu, Seoul, Korea 130-070
TEL:+82-2-584-8959
FAX:+82-2-584-8951