LTCC Chip Antennas – How to maximize performance
Outline

- Chip Antenna Characteristics
- Antenna Selection Considerations
- Circuit Design Constraints
- Layout Tips

Ultimate Goal → To Maximize Performance
Motivation

- Chip Antenna an efficient means of “connectivity” to modern portable compact electronic devices.

- Miniature portable devices requires small antennas.

- Can be internalized – i.e. “Concealed” within device.
Pros

- Chip antennas are small, cheap and performs well.
- Bulky external “whip” type antennas are thing of the past.

Cons

- Must be accounted for during initial circuit design stage
- Interference, proximity de-tuning & degradation concerns.
LTCC Chip Antennas
Chip Antenna Characteristics -1

- Features Ag radiating element encapsulated in ceramic.
- A quarter-wave ($\lambda/4$) monopole system.
- Works with GND plane to form dipole system.
- Certain “No-GND” metal-free space necessary.
- Small form factor, thin profile & light weight
Chip Antenna Characteristics - 2

- Omni-directional radiation.
- Linear Polarization.
- Mounting configuration flexibility.
- Frequency range supported: 0.08 GHz thru 10 GHz.
- WiFi, BT, WiMAX, UWB, GSM, CDMA, GPS etc.
- Suitable for Pick & Place.
Antenna Selection Considerations -1

- Size
- Frequency Band
- Bandwidth
- Polarization
- Peak Gain
- Average Gain
- Radiation Pattern requirements
Antenna Selection Considerations -2

- Successful Antenna design means harmonious interaction of the “seven” parameters (next page)

- Additional considerations for diversity systems
  – e.g. MIMO

- Overall performance is always system dependent.
Circuit Design Constraints

1. Size of the Circuit board.
2. Layout of other board components.
4. Proper GND/No-GND dimensions and clearances.
5. “Tuning” Matching Circuitry
6. Shielding
7. Suitable Enclosure (material)
Layout Tips -1

- Good Placements ①
- Bad Placements ② & ③
Layout Tips -2

Don’t put any metal objects or batteries (if applicable) above or below the yellow region.
Keep away any other metals from clearance area.
Further examples of good antenna placement schemes
Layout Tips -4

- Antenna placement schemes for **antenna diversity** systems

ANT 1
ANT 2
Pattern compensation

Radiation pattern

“Null” Elimination/Compensation
A. Antenna Matching Setup

Calibration Plane of Network Analyzer (NA)

Probe or semi-rigid RF cable

Soldering to: Connect Probe_GND and PCB_GND

50 Ω Feed line

π-type Matching Pads (scheme) preferred

GND

NO GND
B. Measuring Steps

1. One-port (S11) calibration for N.A. (Network Analyzer) Open-Short-Load for desired operating bandwidth

2. Mount probe (semi-rigid RF cable for our example) onto PCB and connect to N.A.

3. Measure S11 of test board **without** antenna or any matching components and save as: $\rightarrow$S11_open $\rightarrow$save trace to memory of N.A.

4. Measure S11 of test board **with antenna and series 0Ω resistor** mounted and save as: $\rightarrow$S11_antenna

5. Set N.A. to data/memory mode (S11_antenna/S11_open) and display/save as: $\rightarrow$S11_match

6. Match the trace of S11_match to **50Ω** (center of Smith chart at the desired frequency)
1. Probe+Feed Line Smith chart display from 1-4GHz (not-normalized)

2. Probe+Feed Line (normalized)

Test Board matching example
1. Probe + Feed Line + Antenna
   Smith chart display from 1-4GHz (not-normalized)

2. Probe + Feed Line + Antenna (normalized)

Test Board matching example
Step 1 in matching:

Ant + shunt 3.9nH (normalized)
Step 2 in matching:

Ant + shunt 3.9nH + series 1.5pF (normalized)

Test Board
matching example

Matched Return Loss chart
Matched Antenna Example

Test Board

EVB p/n: 2450AT45A100-EB1SMA

Return Loss

General Specifications

<table>
<thead>
<tr>
<th>Part Number</th>
<th>2450AT45A100</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency Range</td>
<td>2400 - 2500 Mhz</td>
</tr>
<tr>
<td>Peak Gain</td>
<td>3.0 dBi typ. (XZ-V)</td>
</tr>
<tr>
<td>Average Gain</td>
<td>1.0 dBi typ. (XZ-V)</td>
</tr>
<tr>
<td>Return Loss</td>
<td>9.5 dB min.</td>
</tr>
</tbody>
</table>

b) With Matching Circuit* (wide bandwidth)

* matching circuit and component values will depend on PCB layout, thickness, material, etc.

JTIP/N for Matching Circuit:
Cap (1.5pF): 500R07S1R5BV4T
Inductor (3.9nH): L-07C3N9SV6T
Conclusion – How to design

1\textsuperscript{st} – Determine the antenna location and space available on board

2\textsuperscript{nd} – Select the most appropriate antenna model

3\textsuperscript{rd} - Implement antenna in conformance with design rules

4\textsuperscript{th} – Match antenna to your system