

Security Level:

# LGA Antenna Interfaces Design Guide for ME909u-523 or Later Modules

Version: V2.0

Data: 2014-01-08

[www.huawei.com](http://www.huawei.com)

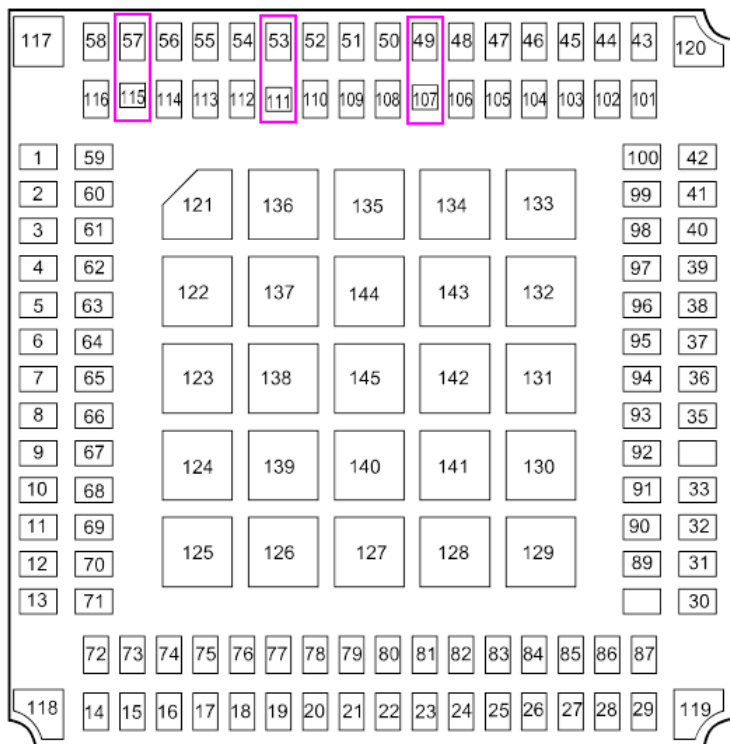
## Revision History

Document Version	Date	Page	Description
1.0	2013-10-17		Creation
2.0	2014-01-08	Page 6	Updated the figure of RF trace on PCB for MU609/ME909u-521
		Page 7	Updated the figure of RF Pad design for MU609/ME909u-521
		Page 8	Updated the Summary -1

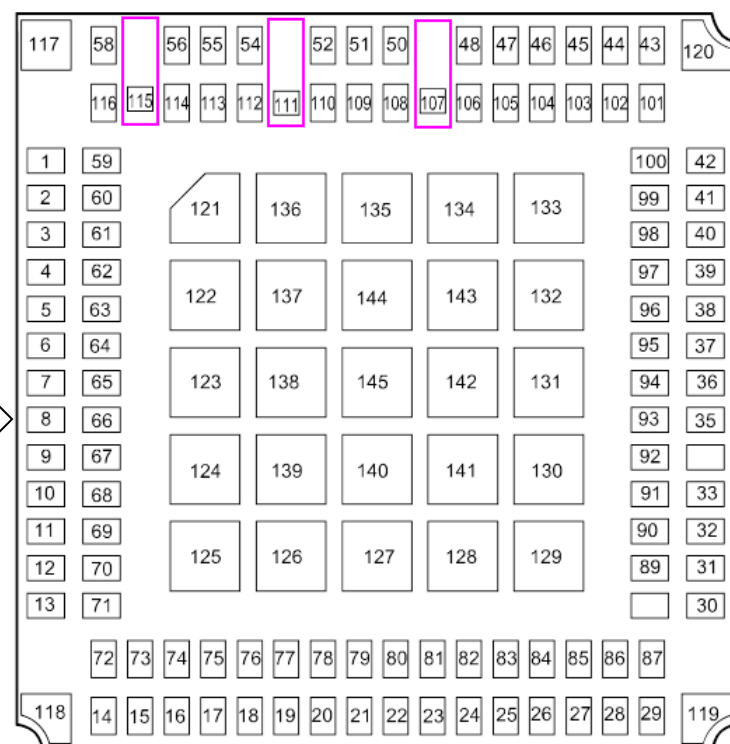
# For 30mm\*30mm LGA Package Module

Interface change point:

MU609/ME909u-521



ME909u-523/later modules



Foot pin is changed.

Top view of LGA interface pins

1. Pin 49, pin 53 and pin 57 are GND.
2. Pin 49, pin 53 and pin 57 should be deleted.
3. Pin 107, pin 111, pin 115 and others are not changed.

These changes can optimize the RF signal trace layout in customers' main board design.

# For Module Migration Design Guide

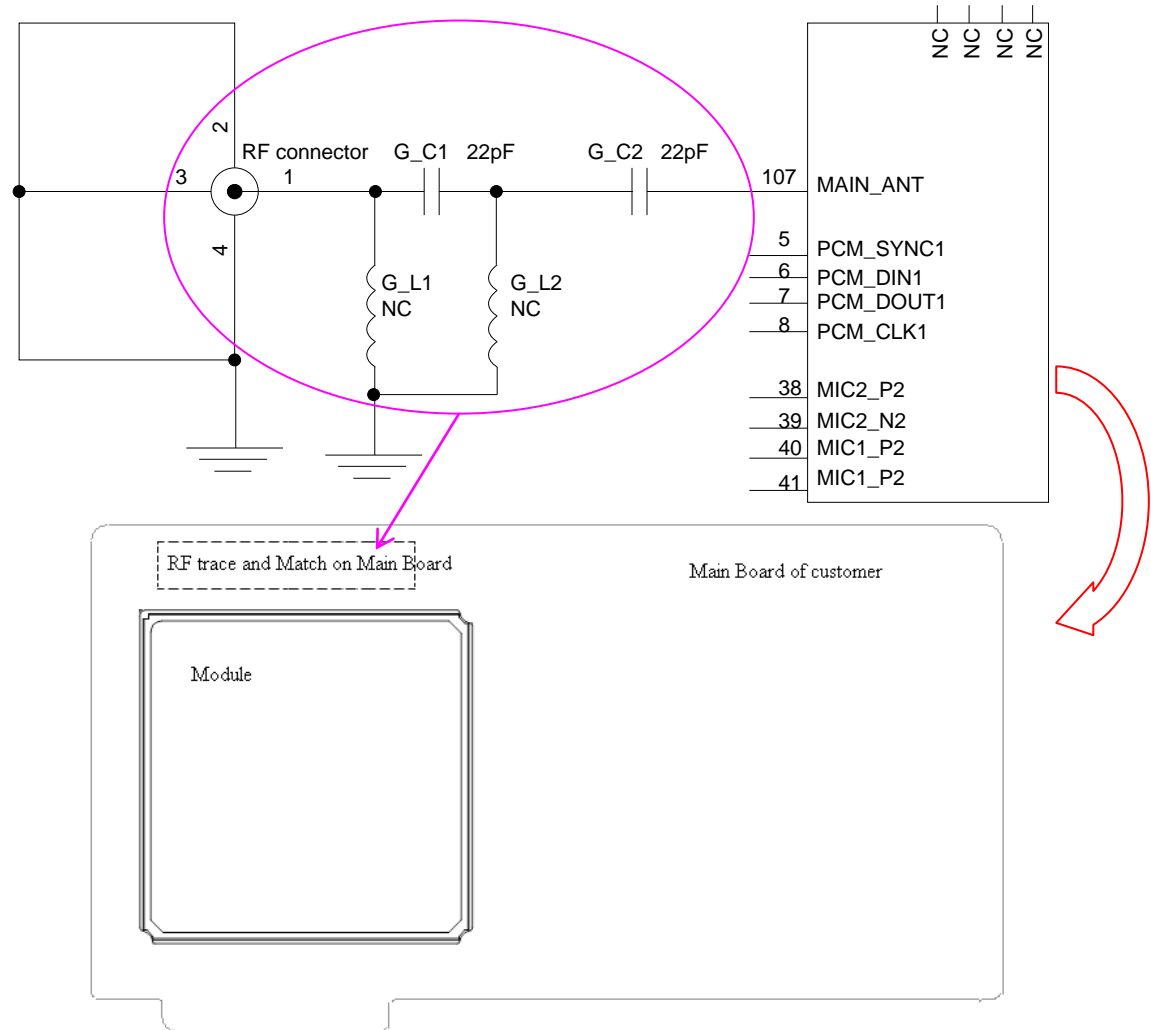
## RF signal trace in the main board design of customers:

pin 107, pin 111 and pin 115 of the module (RF antenna interfaces)

The module is surface mounted on the customer's main board.

In the design of the main board, the RF trace and matching network is necessary in RF antenna interfaces.

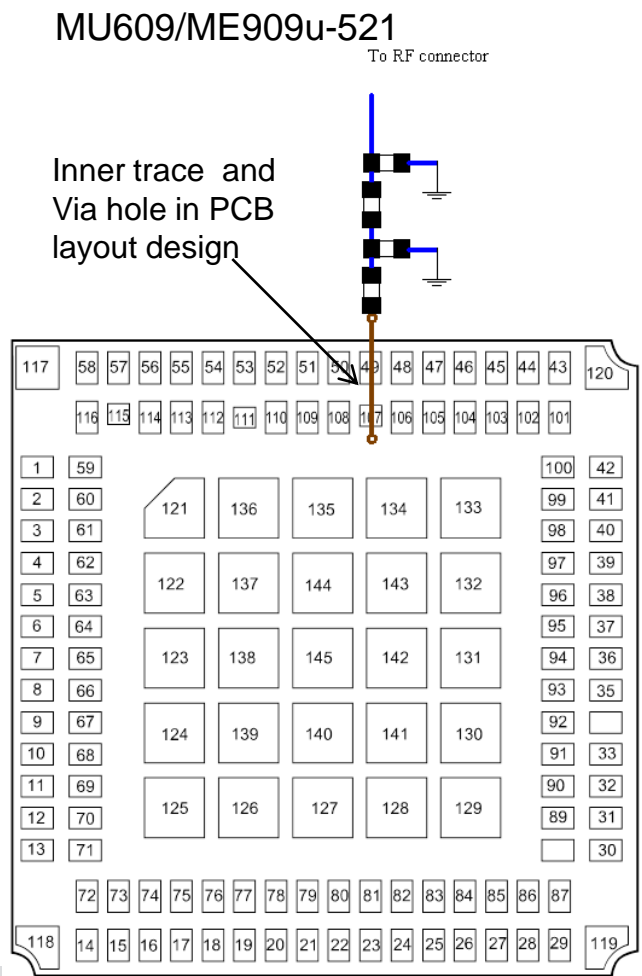
There is a reference schematic with a  $\pi$  type matching network and a DC blocking capacitor.



# For Module Migration Design Guide

**RF signal trace in the main board design of customers:** According to the Interface change point, a new proposal for RF trace layout of antenna interfaces should be considered as shown in following illustration.

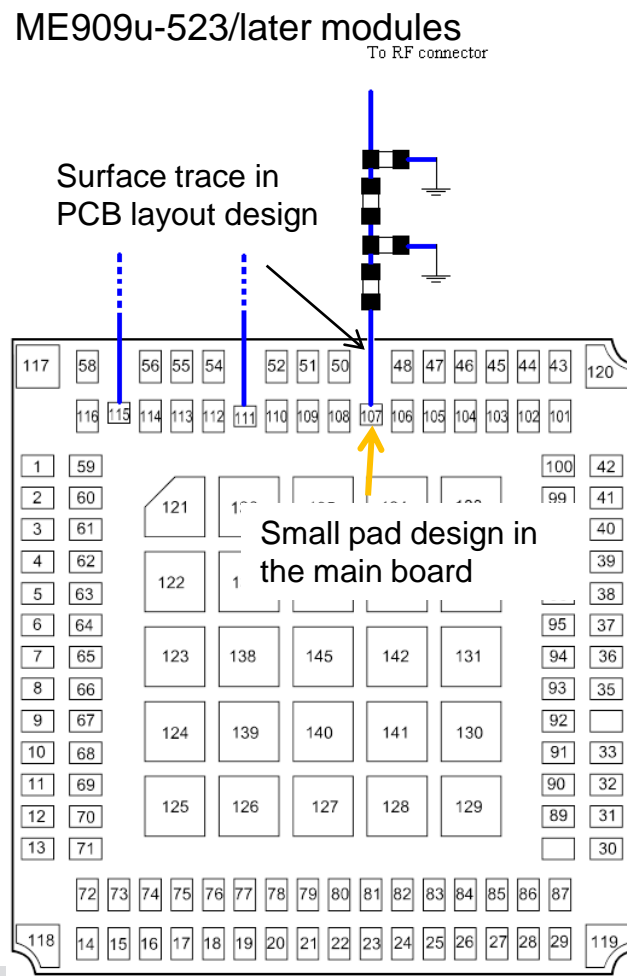
Take RF trace design of pin 107 for example:



RF signal trace in the main board design is changed.

**Recommendation**

Top view of LGA and main board



# For Module Migration Design Guide

## PCB design for RF trace and Pad

### 1. RF trace design

The RF signal (green trace) connects the RF pad and the matching network on the main board. (Because it is the inner trace and via hole in the PCB, there is not shown.) The RF trace impedance needs control to 50 ohm.

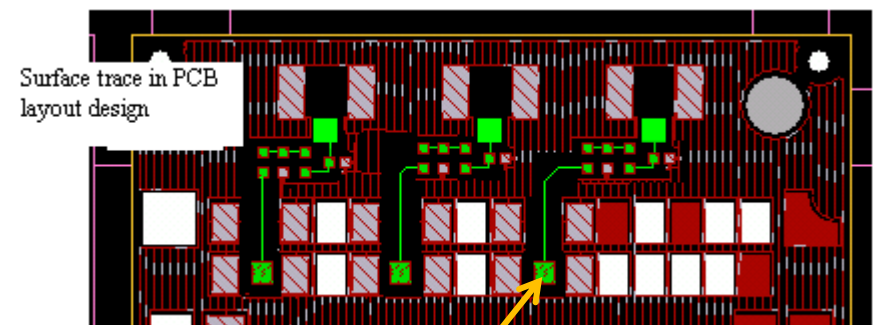
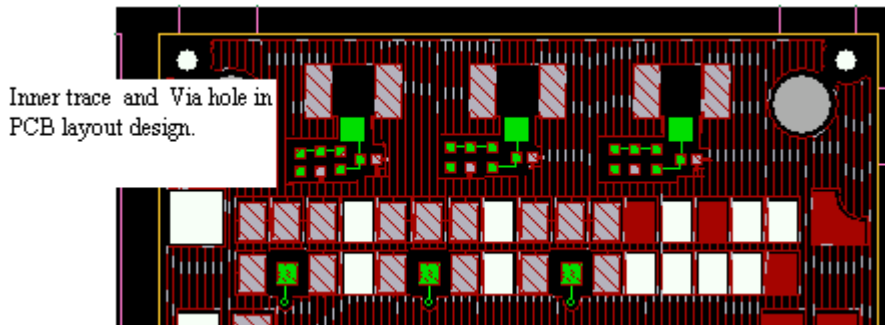
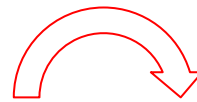
The RF signal (green trace) can be surface trace without via hole on the main board to connect the RF pad and the matching network. The RF trace impedance needs control to 50 ohm.

No corner line by tuning PCB placement can be realized. And the linear trace can have better RF performance.

RF trace on PCB for MU609/ME909u-521

**Recommendation**

RF trace on PCB for ME909u-523/later modules

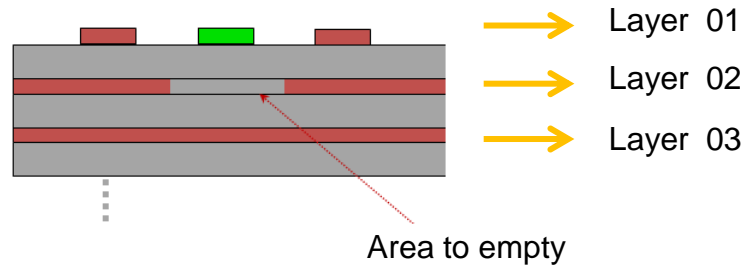


Top PCB layer view of LGA and main board pad

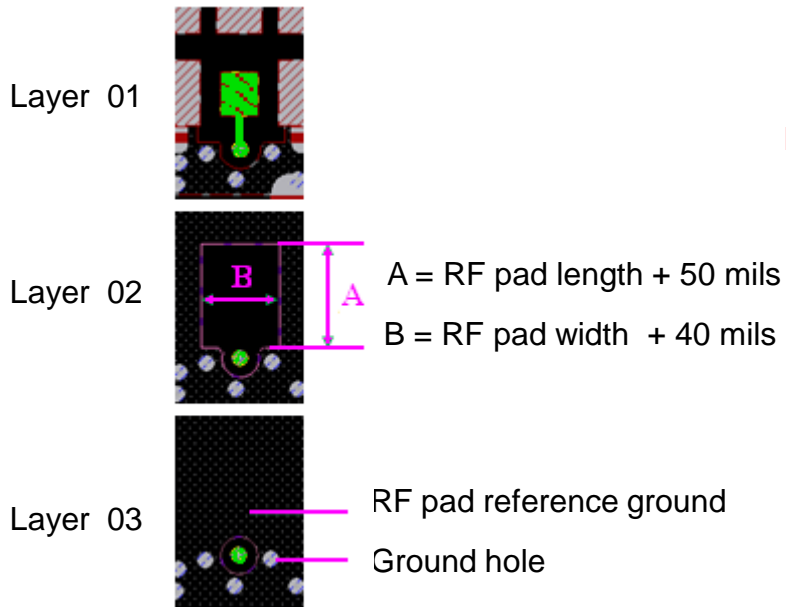
Small pad design in the main board

# For Module Migration Design Guide

## 2. RF Pad design



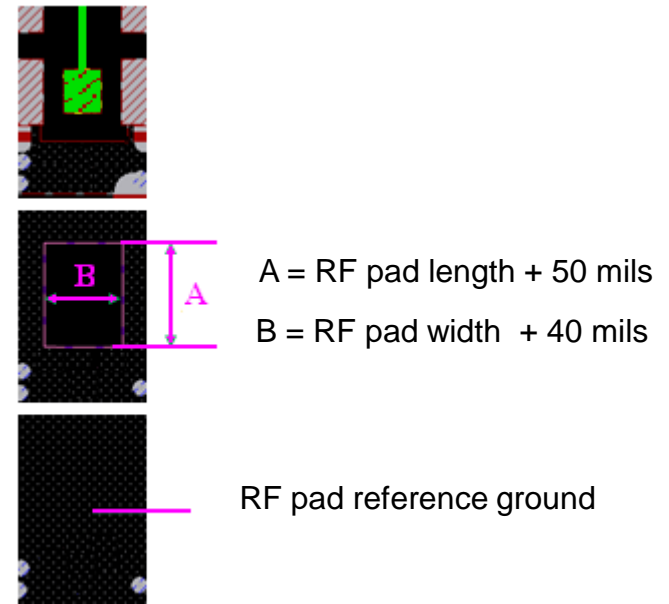
RF Pad design for MU609/ME909u-521



Recommendation



RF Pad design for ME909u-523/later modules



# For Module Migration Design Guide

## Summary

1. RF signal trace in the main board design of customers can be the surface trace in PCB layout design. The RF trace impedance needs control to 50 ohm. The trace should be as short as possible, the match is close to the module.
2. The RF interface pad in the main board PCB should be in small size according to the module's pin size. And we recommend to dig the adjacent layer below the RF pad.
3. The changed design can have better RF performance for customers: reducing the insert loss, VSWR and improving the RF power/sensitivity on low/high bands.
4. The pin 111 and pin 115 signal trace layout can be the same as pin 107 (MAIN\_ANT) .

Huawei recommends that in the migration of ME909u-523 and later modules, **RF signal trace should be surface trace and with the small pad in PCB layout design for customers' main board.**



# For Module Migration Design Guide

## The effect of change:

1. The migration of old and new modules in customers' design:

The effect of change	Old module package (MU609/ME909u-521)	New module package (ME909-523)
Old main board, Inner trace and Via hole in PCB layout design	Design guide as before. Pay attention to RF Pad design requirement of MU509 series modules.	Design guide as before. Change pins have less effect on customer's main board design.
New main board, surface trace in PCB layout design	<b>Not recommended, need confirm in real PCB board.</b>	Recommended

2. Better insert loss and VSWR in PCB simulation. Frequency range: 700 MHz~2.7 GHz

Insert loss/VSWR	Old module package (MU609/ME909u-521)	New module package (ME909-523)
Old main board, Inner trace and Via hole in PCB layout design	IL: 0.35 @2.7GHz VSWR: 1.34 @2.7GHz	IL: 0.35 @2.7GHz VSWR: 1.34 @2.7GHz
New main board, surface trace in PCB layout design	<b>Not recommended, VSWR is not good by tuning match.</b>	IL: 0.1~0.27 dB(700MHz~2.7GHz) VSWR: 1.1~1.3(700MHz~2.7GHz)

## Summary

1. The package of 30mm\*30mm LGA modules will be changed in ME909u-523 and later modules.
2. PCB design for RF trace and pad on customers' main board should be adapted to this updated one.
3. Huawei recommends RF signal trace should be surface trace and with the small pad in PCB layout design for customers' main board.

# Thank you

[www.huawei.com](http://www.huawei.com)

**Copyright©2014 Huawei Technologies Co., Ltd. All Rights Reserved.**

The information in this document may contain predictive statements including, without limitation, statements regarding the future financial and operating results, future product portfolio, new technology, etc. There are a number of factors that could cause actual results and developments to differ materially from those expressed or implied in the predictive statements. Therefore, such information is provided for reference purpose only and constitutes neither an offer nor an acceptance. Huawei may change the information at any time without notice.