

# **AW-CM572-SUR**

**IEEE 802.11 a/b/g/n/ac WLAN 2T2R with  
Bluetooth 5.0 Combo LGA Module**

## **Layout Guide**

**Rev. A**

## Revision History

Version	Revision Date	Description	Initials	Approved
A	2022/8/17	● Initial Version	Jeff Kuo	N.C. Chen

## 1. INTRODUCTION

This document provides key guidelines and recommendations to be followed when creating AW-CM572-SUR layout. It is strongly recommended that layout be reviewed by AzureWave engineering team before released for fabrication.

The following is a summary of the major items that are covered in detail in this application note. Each of these areas of the layout should be carefully reviewed against the provided recommendations before the PCB goes to fabrication.

- Ground Layout
- Power Layout
- Digital Interface
- RF Trace
- The other layout guide Information

## 2. Ground Layout

Please follow general power layout guidelines. Here are some general rules for customers' reference.

- (1) The top layer of customer platform should keep **complete ground plane as possible as you can, in order to be connected for all ground pins of AW-CM572-SUR module.**
- (2) The area under our module forbidden any trace and via on top layer of customer platform.

## 3. Power Layout

Please follow general power layout guidelines. Here are some general rules for customers' reference.

- (1) Power traces shall surround ground to get stable and make sure all power traces have good return path to ground.
- (2) Do not get close to digital traces (SDIO, UART) or continuous data traces, there could be coupling noise affect power traces and IC.

## 4. Digital Interface

Please follow power and ground layout guidelines. Here are some general rules for customers' reference.

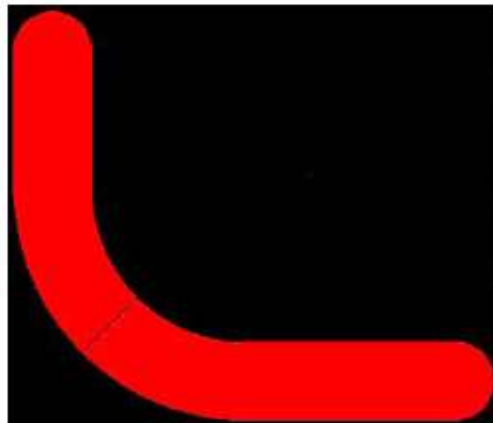
- (1) The digital Interface to the module must be well routed to minimize coupling to power planes and other digital signals.
- (2) SDIO and UART Traces need GND surrounded.
- (3) SDIO pin can add pull high resistor on traces close WIFI module.(reserved solution)
- (4) SDIO and UART traces between host and module are shot as possible.
- (5) SDIO Traces length between host and module is equal as possible.
- (6) SDIO & UART traces as possible away from CLOCK signal

## 5. RF Trace

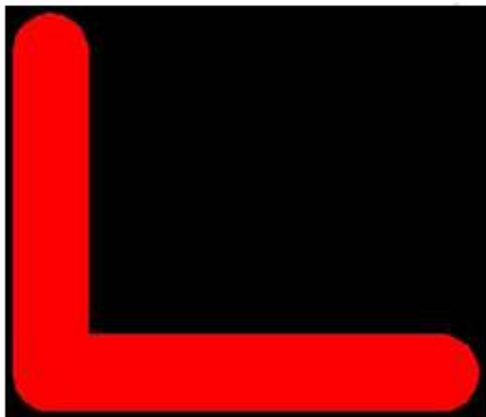
The RF trace is the critical to route. Here are some general rules for customers' reference.

- (1) The RF trace impedance should be  $50\Omega$  between ANT port and antenna matching network.
- (2) The length of the RF trace should be minimized.
- (3) To reduce the signal loss, RF trace should laid on the top of PCB and avoid any via on it.
- (4) The CPW (coplanar waveguide) design and the microstrip line are both recommended; the customers can choose either one depending on the PCB stack of their products.
- (5) The RF trace must be isolated with a ground beneath it. Other signal traces should be isolated from the RF trace either by ground plane or ground vias to avoid coupling.
- (6) To minimize the parasitic capacitance related to the corner of the RF trace, the right angle corner is not recommended.

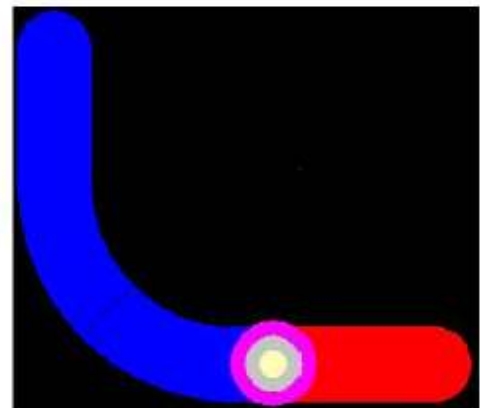
If the customers have any problem in calculation of trace impedance, please contact Azurewave.



**Correct RF trace**



**Right-angled corner**



**Via on RF trace**

### **Incorrect RF trace**

The information contained herein is the exclusive property of Azurewave and shall not be distributed, reproduced, or disclosed in whole or in part without prior written permission of AzureWave.

## **6. The other layout guide Information**

1. Keep the module unused function pins floating.
2. High speed interface (i.e. UART/SDIO/HSIC) shall have equal electrical length. Keep them away from noise sensitive blocks.
3. Good power integrity of VDD will improve the signal integrity of digital interfaces.
4. Clock (SUSCLK) should have complete ground to make sure coupling will not happen to any other path.

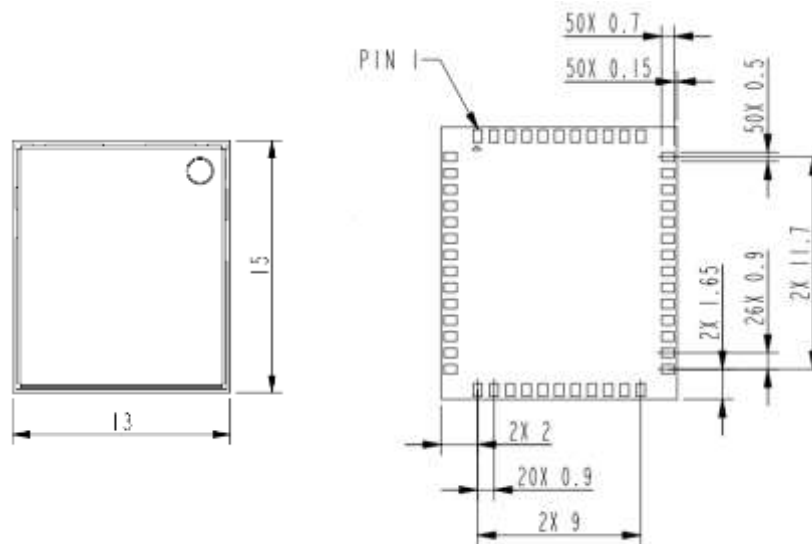
## 7. Footprint and stencil aperture recommendation

Footprint shares the same center with pin pad land, and follows below rule to define the size.

1. Rectangle type:
  - 1.1 Outermost layer :
 

Footprint length = 1 \* (pin pad land length)

Footprint width = 1 \* (pin pad land width)
  - 1.2 Others: the same size with pin pad.
2. Stencil aperture suggestion
  - 2.1 The pad size is same size with footprint.
3. Stencil thickness suggestion:
  - 3.1 Normal type product thickness: 0.1mm~0.12mm.
4. Solder printer opening and customer PCB footprint suggest.



## 8. Mechanical Drawing

