

Date: 2023-08-14

## Declaration for KDB 996369 Module Q&A

FCC ID: [TLZ-AM497617](#)

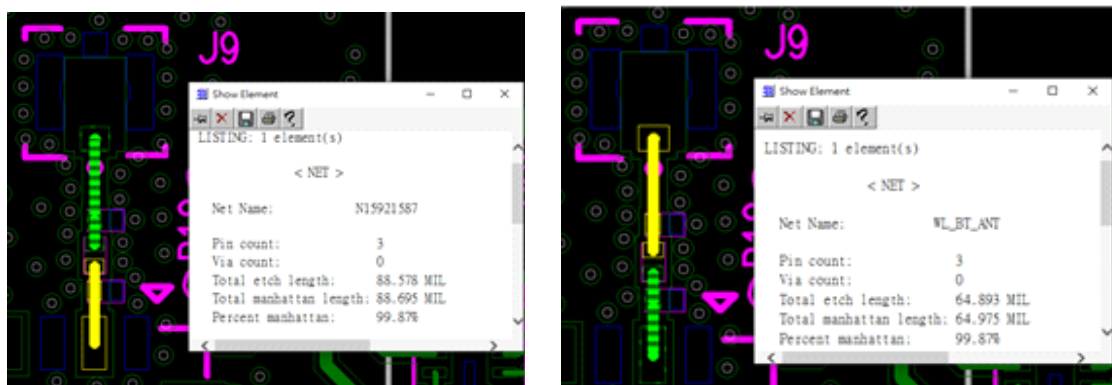
To whom it may concern,

Question 1. Information that includes permitted variances(e.g., trace boundary limits, thickness, length, width, shape(s), dielectric constant, and impedance as applicable for each type of antenna);

**Description:**

The AW-AM497 is configured for monostatic operation, which requires only a single RF I/O pin for full simplex communication. The output must be routed to the antenna via 50 ohm microstrip or stripline on the PCB. No coupling capacitor is required given that the RF pin is AC-coupled internal to the AW-AM497. To maximize the radiated power (and corresponding communication range), the length of the transmission line between \_Model name\_ and antenna should be made as short as possible.

The AW-AM497 is designed for 50 Ohm characteristic impedance, and the connection between AW-AM497 and antenna should all be designed for 50 Ohm characteristic impedance. If using PCB connections, should design the PCB layout for 50 Ohm characteristic impedance using microstrip, stripline, etc. (Figure 1)



Microstrip

Figure 1: PCB Transmission Line Styles

For example, user can design Microstrip according to PCB stack up (Figure 2) by tool Polar (Microstrip Impedance, Figure 3) to get PCB path width for RF is 15 mil.

| Layer           | Layer Type | Layer Structure | Thickness(mil) |
|-----------------|------------|-----------------|----------------|
|                 | Soldermask |                 | 0.8            |
| L1              | Top        | 0.5oz+Plating   | 1.7            |
|                 |            | Prepreg         | 5              |
| L2              | Plane      | 1oz             | 1.26           |
|                 |            | Core            | 12             |
| L3              | Plane      | 1oz             | 1.26           |
|                 |            | Prepreg         | 5              |
| L4              | Bottom     | 0.5oz+Plating   | 1.7            |
|                 | Soldermask |                 | 0.8            |
| Total Thickness |            |                 | 29.52          |

Figure 2: PCB Stack Up

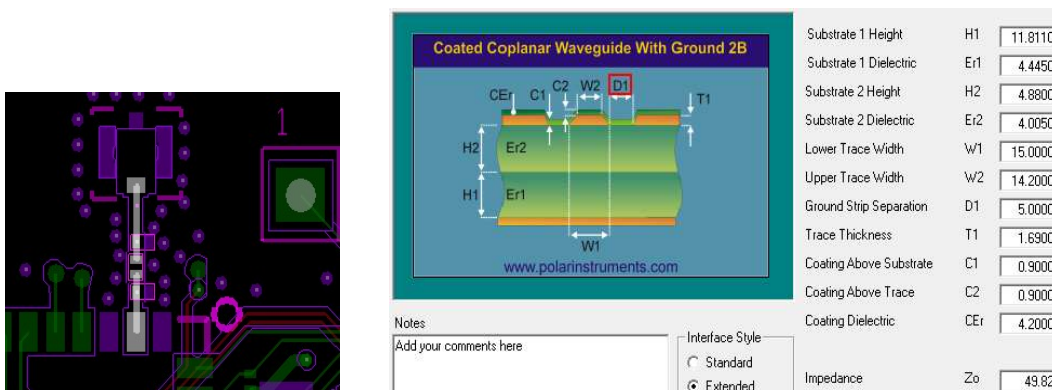


Figure 3: Microstrip Impedance

Question 2. Each design shall be considered a different type (e.g., antenna length in multiple(s) of frequency, the wavelength, and antenna shape (traces in phase) can affect antenna gain and must be considered);

### Description:

If, as recommended above, a microstrip trace is used as the RF transmission line, there are additional guidelines that can be applied to minimize interference with other signals on the PCB. Most notable are clearance between the RF traces and other nearby traces, and via stitching to connect ground fills to one another.

This section provides a bit of detail.

Any conductor carrying electrical current can potentially act as an antenna, 2400~2500 MHz frequency. In order to minimize interference between traces, they should be adequately spaced.



AzureWave Technologies, Inc.

8F., No.94, Baozhong Rd., Xindian Dist., New Taipei City 23144, Taiwan

TEL: +886-2-55995599

FAX: +886-2-66289666

Ideally, all RF traces should be surrounded on the same layer by ground fill.

Question 3. The parameters shall be provided in a manner permitting host manufacturers to design the printed circuit (PC) board layout;

**Description:**

I In order to operate FCC ID: TLZ-AM497617 the product must strictly use only with the following antennas or antenna types with maximum gain as shown.

MAG. LAYERS Scientific-Technics Corporation., LTD model MSA-4008-25GC1-A2 Linear Vertical polarization antenna with 2.98dBi (2.4~2.5GHz) and 5.16dBi (4.9~5.9GHz) gain.

Question 4. Appropriate parts by manufacturer and specifications.

**Description:**

Reference question 1

Question 5. Test procedures for design verification.

**Description:**

RF I/O interface to antenna connector on the PCB shall accomplished via microstrip MHF4 connector. The multiplexer on carry board PCB with interface to antenna.

The connector on carry board PCB with interfaces to antenna must be of a unique type to disable connection to a non-permissible antenna in compliance with FCC section 15.203. The following connectors are allowed.

1. MHF4 Connector : I-PEX, model 20449-001E or equivalent

Custom 50 ohm coaxial pigtail from PCB to antenna

Question 6. Production test procedures for ensuring compliance.

**Description:**

AW-AM497 shall follow the above design rule for characteristic impedance of 50 ohms +/- 10% and measure microstrip or stripline transmission line width on PCB.

Question 7. Production test procedures for ensuring compliance.

**Description:**

AW-AM497 shall measure output power to make sure the tolerance of output power meet AW-AM497 specification. It proofs the characteristic impedance is 50 ohms +/- 10%.



AzureWave Technologies, Inc.

8F., No.94, Baozhong Rd., Xindian Dist., New Taipei City 23144, Taiwan

TEL: +886-2-55995599

FAX: +886-2-66289666

Thank you for your attention.

Sincerely yours,

A handwritten signature in black ink, appearing to be 'Patrick Lin', is written over a horizontal dashed line.

Patrick Lin / Director

**AzureWave Technologies, Inc.**

Tel: 02-55995599 \*5580

Fax: 02-66289666

E-mail: [patrick.lin@azurewave.com](mailto:patrick.lin@azurewave.com)