Date: 2024-05-17

Declaration for KDB 996369 Module Q&A

FCC ID: N6C-IM100

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:

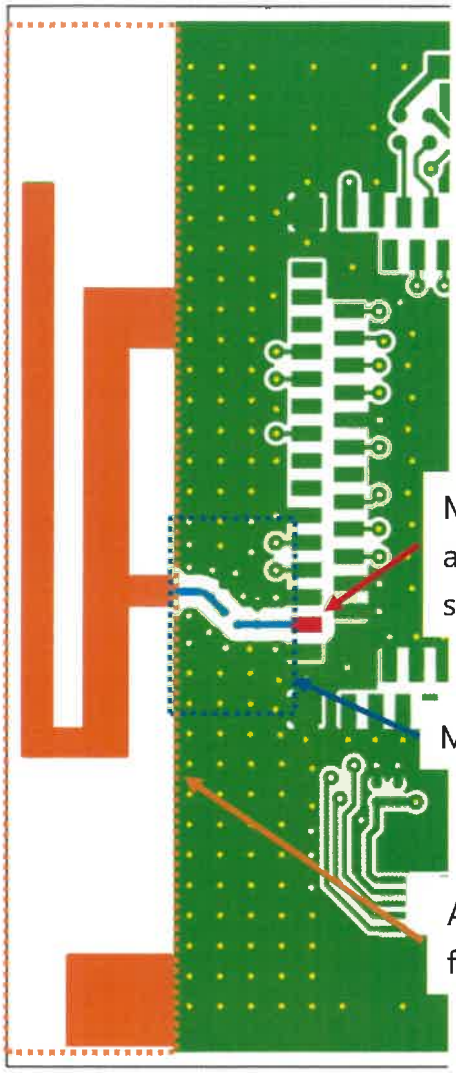
IM-100 is configured for monostatic operation, which requires only a single RF I/O pin for full simplex communication. The output must be routed to the antenna via 50 ohm microstrip or stripline on the PCB. No coupling capacitor is required given that the RF pin is AC-coupled internal to IM-100. To maximize the radiated power (and corresponding communication range), the length of the transmission line between IM-100 and antenna should be made as short as possible.

IM-100 is designed for 50 Ohm characteristic impedance, and the connection between IM-100 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.

For example, user can design Microstrip according to PCB stack up by tool Polar (Microstrip Impedance) to get PCB path width for RF is XX mm.

		thickness
	SolderResist	30um
L1	Metal	42um
	FR-4	110um
L2	Metal	35um
	FR-4	1130um
L3	Metal	35um
	FR-4	110um
L4	Metal	42um
	SolderResist	30um

Total thickness = 1.6mm



Material of PC board:

PCB material: FR-4 (ε_r = 4.25-4.35 @1GHz)

Metal thickness: 42um +/- 10um

Total thickness: 1.6mm +/- 0.2mm

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 on 2400~2500 MHz and 5150~5850MHz frequency. In order to minimize interference between traces, they should be adequately spaced.

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:

In order to operate FCC ID: **N6C-IM100** the product must strictly use only with the following antennas or antenna types with maximum gain as shown.

Brand	Model	Antenna Net Gain(dBi)	Frequency range	Antenna Type	Connector Type
Silex Technology	SXANTFDB24A55-03	2.75	2.4-2.4835GHz	Folded inverted-L	None (On-board)
		1.82	5.15-5.25GHz		
		1.82	5.25-5.35GHz		
		2.82	5.47-5.725GHz		
		2.99	5.725-5.80GHz		

Question 4. Appropriate parts by manufacturer and specifications.

Description:

Refer to Question 1.

Question 5. Test procedures for design verification.

Description:

RF I/O interface to antenna connector on the PCB shall accomplished via microstrip MHF1 connector. 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.

MHF1 Connector: I-PEX, model 20279-001E-03 or equivalent Custom 50 ohm coaxial connector

Question 6. Production test procedures for ensuring compliance.

Description:

IM-100 shall measure output power to make sure the tolerance of output power meet IM-100 specification. It proofs the characteristic impedance is 50 ohms +/- 10%.

Thank you for your attention.

Sincerely yours,

Yukinari Shibuya, Manager
Silex Technology, Inc.
Tel: +81-774-98-3878
Fax: +81-774-98-3758
E-mail: shibuya@silex.jp