BTUC49-TX&SP Trace Layout

13/12/18

For proper integration of the BTUC49-TX&SP module in end products the following requirements must be met. The BTUC49-TX&SP modules have same hardware and According to end product, BTUC49-TX&SP will be mounted at two different board, which has same type antenna and similar RF trace.
A host product incorporating the BTUC49-TX&SP modules cannot take advantage of the pre-existing certification of the component transmitter without conformity to the specific requirements in these instructions.

1. Contents

2. Approved RF Connection (BTUC49-TX)

This is the preferred RF connection design and is approved with all operating modes of the module.
This design uses a PCB microstrip to connect the BTUC49-TX module's Antenna
A 2 layer PCB is used in this design. The Top layer 16mil RF trace and bottom ground plane form an RF microstrip.

- Figure 1 shows the required layer stackup for this design and must be matched precisely including material type, dielectric thickness, and copper thickness.
- Figure 2 shows the trace dimensions that must be followed precisely, including trace width, routing, and RF matching components. A 0ohm resistor (TA-I 1R-0000000-J200) is located at Pi-matching by series connect, between the module and RF switch. A RF switch is located at Between antenna and Pi-matching which is only for connecting RF connector to RF testing.
- The Ground plane on bottom must not have any cutouts in the area under the RF trace, in the area between the module and the antenna, or under the RF matching components.

Layer	Layer Name	Material	Nominal Thickness (mil)
	Top soldermask		0.40
1	1/2oz + Plating		1.4
	core(36mil H/H)		34.6
2	1/2oz + Plating		1.4
	Bottom soldermask		0.4

Figure 1 - PCB Stackup

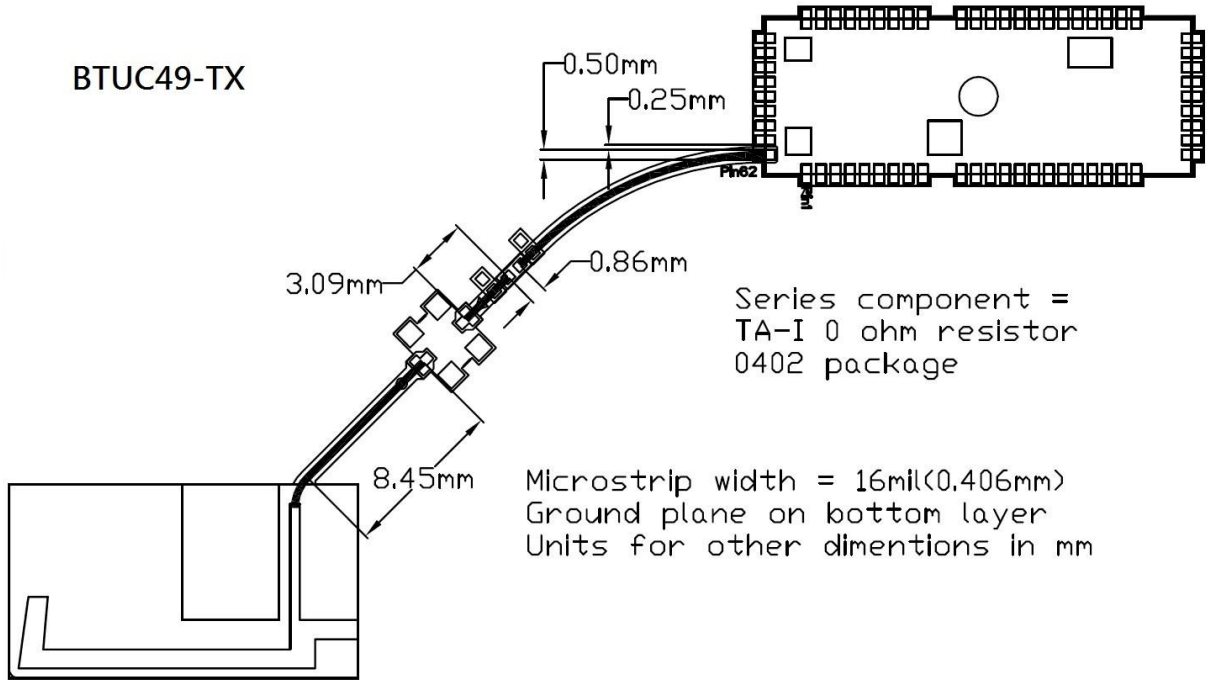

Figure 2 - Trace Dimensions and RF Matching Component Placement for BTU49-TX

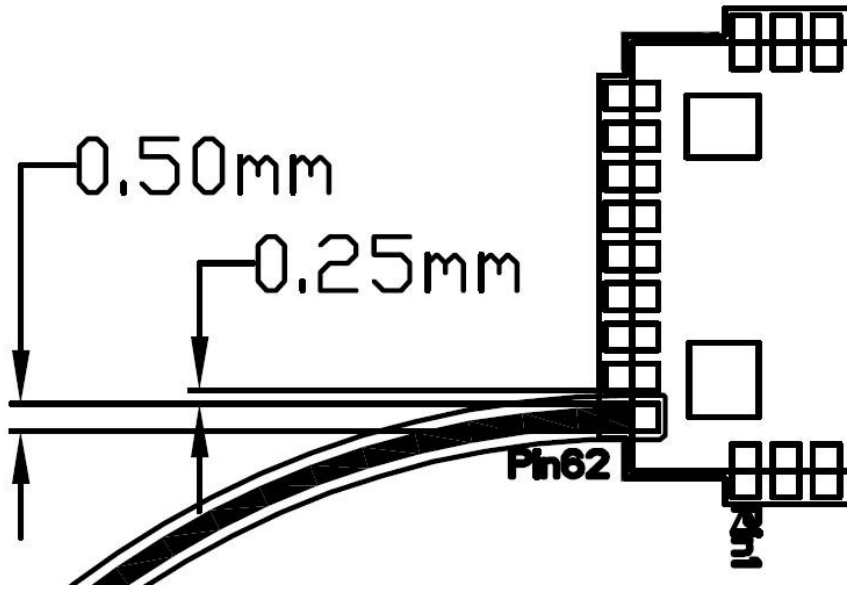

Figure3 – Pin size for BTUC49-TX

3. Alternate RF Connection (BTUC49-SP)

This is the preferred RF connection design and is approved with all operating modes of the module.
This design uses a PCB microstrip to connect the BTUC49-SP module's Antenna.
The RF trace's difference for TX and SP is the path between antenna feed point and RF switch only.
A 2 layer PCB is used in this design. The Top layer 16mil RF trace and bottom ground plane form an RF microstrip.

- Figure 4 shows the required layer stackup for this design and must be matched precisely including material type, dielectric thickness, and copper thickness.
- Figure 5 shows the trace dimensions that must be followed precisely, including trace width, routing, and RF matching components. A 0ohm resistor (TA-I 1R-0000000-J200) is located at Pi-matching by series connect, between the module and RF switch. A RF switch is located at Between antenna and Pi-matching which is only for connecting RF connector to RF testing.
- The Ground plane on bottom must not have any cutouts in the area under the RF trace, in the area between the module and the antenna, or under the RF matching components.

Layer	Layer Name	Material	Nominal Thickness (mil)
	Top soldermask		0.40
1	1/2oz + Plating		1.4
	core(36mil H/H)		34.6
2	1/2oz + Plating		1.4
	Bottom soldermask		0.4

Figure 4 - PCB Stackup

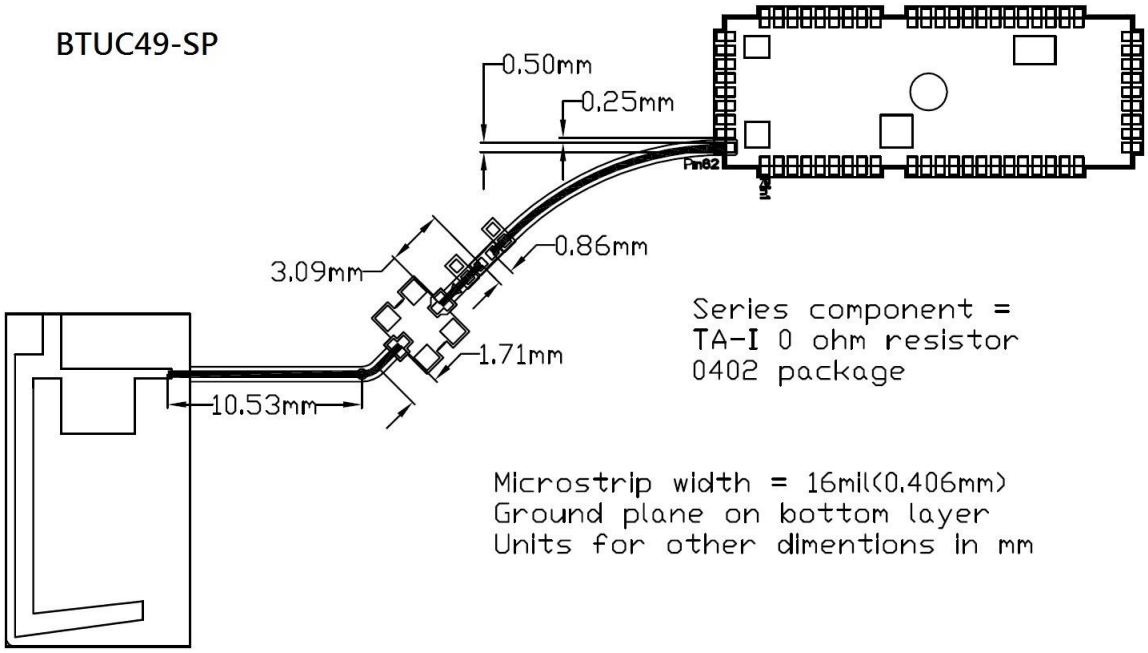

Figure 4 - Trace Dimensions and RF Matching Component Placement for BTU49-SP

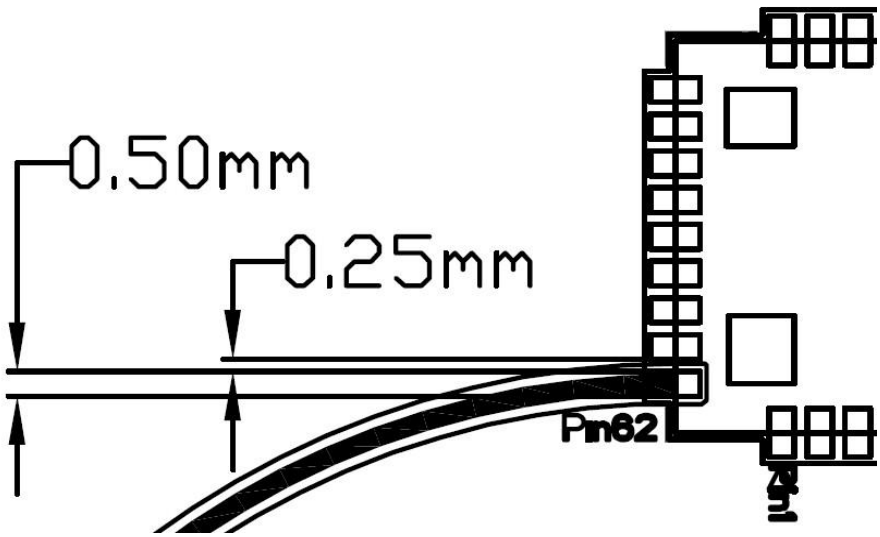

Figure5 – Pin size for BTUC49-SP

4. Approved Antennas

The antenna in Table 1 is tested and approved with BTUC49-TX. According to the FCC Permissive Change Policy (178919 D01)Additional" antennas that are equivalent may be substituted, and then marketed without a Class II permissive change…Equivalent antennas must be of the same type (e.g., yagi, dish, etc.), must be of equal or less gain than an antenna previously authorized under the same grant of certification (FCC ID), and must have similar in-band and out-of-band characteristics. The antenna in Table 2 is alternate antenna which is follow the FCC Permissive Change Policy (178919 D01) and is used for connecting to BTUC49-SP module.

Manufacturer	Part number	Type	Peak Gain	Module Type
N/A	N/A	PIFA	2.49dBi	TX

Table 1 - Approved Antennas

Manufacturer	Part number	Type	Peak Gain	Module Type
N/A	N/A	PIFA	2.4dBi	SP

Table 2 - Alternate Antennas

5. Design Verification Test Procedures

After the design is fabricated the following measurements should be executed to verify the design:

1. Mechanical measurement of dimensions specified in the Microstrip Dimensions diagrams above
2. Obtain and review the detailed layer stackup solution used for the build from the PCB manufacturer that specifies dielectric thicknesses and target dielectric constants for substrate materials.

6. Production Test Procedures for Ensuring Compliance

During production test for the host device, The BTUC49-TX&SP module are to be activated in maximum power transmit and the conducted RF output power at RF switch is to be measured using a Spectrum Analyzer, RF Power Meter or other appropriate RF measurement equipment. The conducted output power should not exceed the maximum output power specified in the BTUC49-TX&SP Data Guide.