Linx TECHNOLOGIES is now part of TE connectivity

Amplified HumPRO™ Series RF Transceiver PCB Layout Guide

Reference Guide RG-00110

Introduction

The Amplified HumPRO™ Series RF transceiver module has obtained a modular approval from the United States FCC and Industry Canada. Products that wish to make use of this modular certification must meet specific PCB layout and implementation requirements. These requirements are outlined in this guide.

Approved RF Connections for HUM-A-900-PRO-UFL

The HUM-A-900-PRO-UFL module employs an integrated u.FL RF connector. Therefore, the design and layer stack up of the host PCB does not need to meet any special requirements to maintain the validity of the module's pre-certified status. An approved cable assembly and antenna are required and are specified in the Approved Antennas section.

Approved RF Connections for HUM-A-900-PRO-CAS

The HUM-A-900-PRO-CAS outputs the RF onto a castellation on the module. This allows a designer to route the RF to a different connector or PCB antenna. However, the product's PCB now becomes part of the RF design and must, therefore, match the design that was used when the module was tested.

Edge Mount RP-SMA Connection

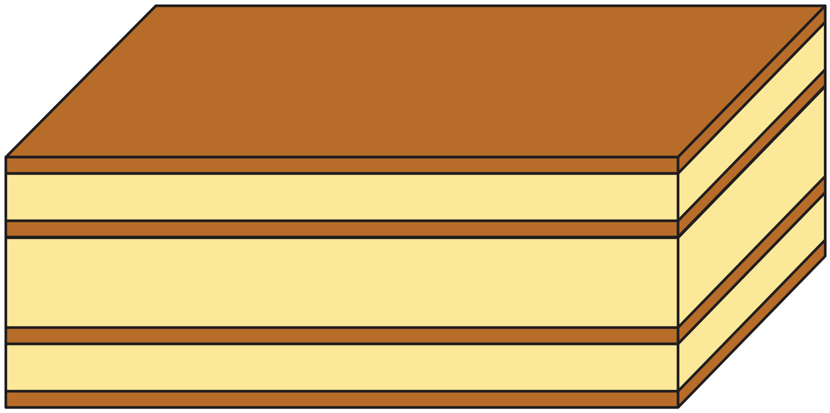
This RF connection design is approved with all operating modes of the module. This design uses a PCB microstrip to connect the module's antenna castellation to an edge mount reverse polarity SMA connector (Linx part number CONREVSMA003.062). A four-layer PCB is used in this design. A microstrip trace is formed by the 24mil wide trace on the top layer and the ground plane on Mid-Layer 1.

Figure 1 shows the required PCB layer stack. This layer stack up must be matched precisely including material type, dielectric constant, dielectric thickness, and copper thickness.

Figure 2 shows the trace dimensions that must be followed precisely, including trace width, and routing.

The Ground plane on Mid-layer 1 must not have any cutouts in the area under the RF trace or the area between the module and the connector.

Other components required by the application can be placed around the module. The ground plane can be used as the common ground for other circuits on the board. The general board layout guidelines should also be followed.



Layer Name	Thickness	Material
Top Layer	1.4mil	Copper
Dielectric 1	14.0mil	FR-4 (Er = 4.6)
Mid-Layer 1	1.4mil	Copper
Dielectric 2	28.0mil	FR-4 (Er = 4.6)
Mid-Layer 2	1.4mil	Copper
Dielectric 3	14.0mil	FR-4 (Er = 4.6)
Bottom Layer	1.4mil	Copper

Figure 1: HUM-A-900-CAS with an Edge Mount RP-SMA Connector PCB Layer Stack

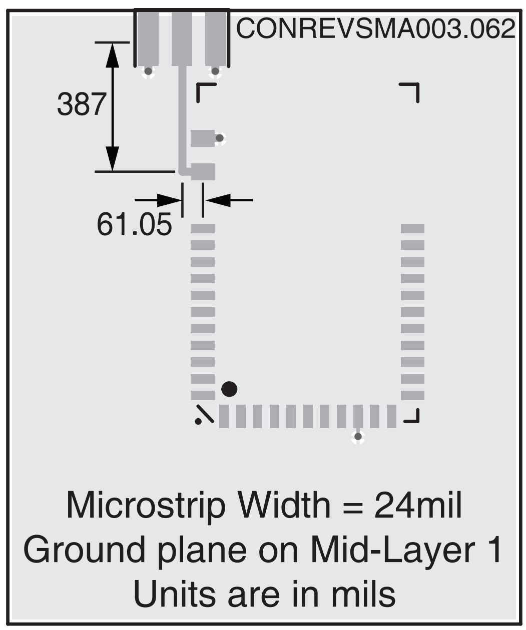


Figure 2: HUM-A-900-CAS with an Edge Mount RP-SMA Connector PCB Layout

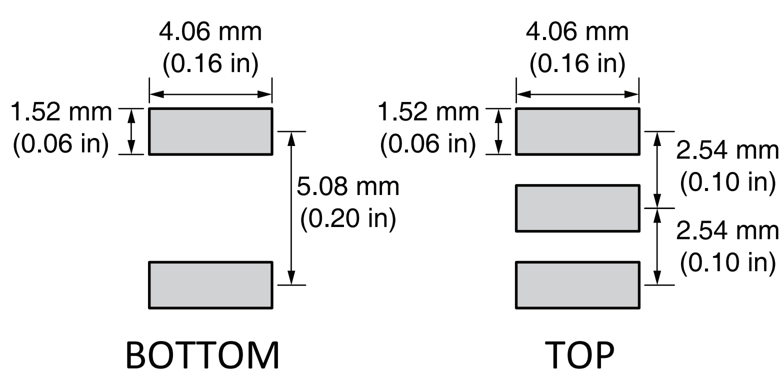


Figure 3: CONREVSMA003.062 PCB Footprint

MicroSplatch Antenna Connection

This RF connection is approved with all operating modes of the module. This design uses a PCB microstrip to connect the module's Antenna castellation to the MicroSplatch surface mount antenna (Linx part number ANT-916-USP). A four-layer PCB is used in this design. A microstrip trace is formed by the 24mil wide trace on the top layer and the ground plane on Mid-Layer 1.

Figure 4 shows the required PCB layer stack. This layer stack up must be matched precisely including material type, dielectric constant, dielectric thickness, and copper thickness.

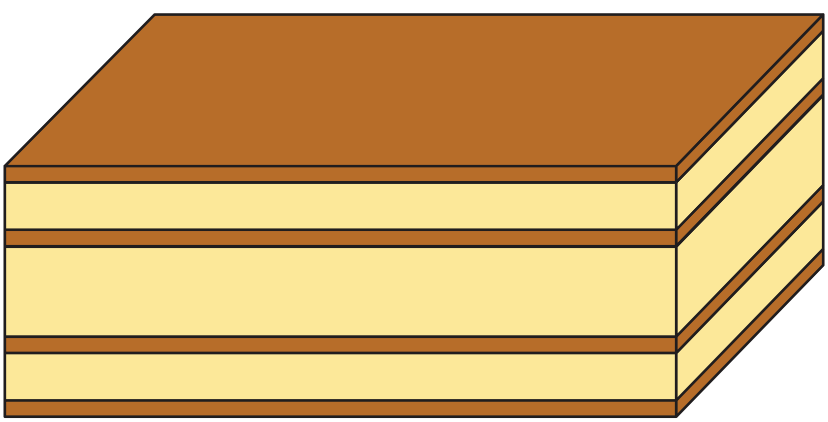
Figure 5 shows the trace dimensions that must be followed precisely, including trace width, and routing.

The Ground plane on Mid-layer 1 must not have any cutouts in the area under the RF trace or the area between the module and the antenna.

Other components required by the application can be placed around the module. The ground plane can be used as the common ground for other circuits on the board. The general board layout guidelines should also be followed.

Figure 6 shows the dimensions of the test fixture of the ANT-916-USP antenna. This antenna is a quarter-wave monopole, which is half of the antenna structure. The other half is the ground plane on the PCB. Without going into the details, the size of this ground plane is important. Anything smaller will shift the antenna's resonant frequency higher and could push it out of the band if it is too small. Anything larger will shift the antenna's frequency lower, but the amount of the shift will not be as much as what is caused by a smaller ground plane. It is better to have too much plane than not enough.

There are no traces, planes or any copper under the antenna or to its sides on any layer of the board. Anything conductive in this area will impact the radiation pattern and the antenna's performance.



Layer Name	Thickness	Material
Top Layer	1.4mil	Copper
Dielectric 1	14.0mil	FR-4 (Er = 4.6)
Mid-Layer 1	1.4mil	Copper
Dielectric 2	28.0mil	FR-4 (Er = 4.6)
Mid-Layer 2	1.4mil	Copper
Dielectric 3	14.0mil	FR-4 (Er = 4.6)
Bottom Layer	1.4mil	Copper

Figure 4: HUM-A-900-CAS with the MicroSplatch Antenna PCB Layer Stack

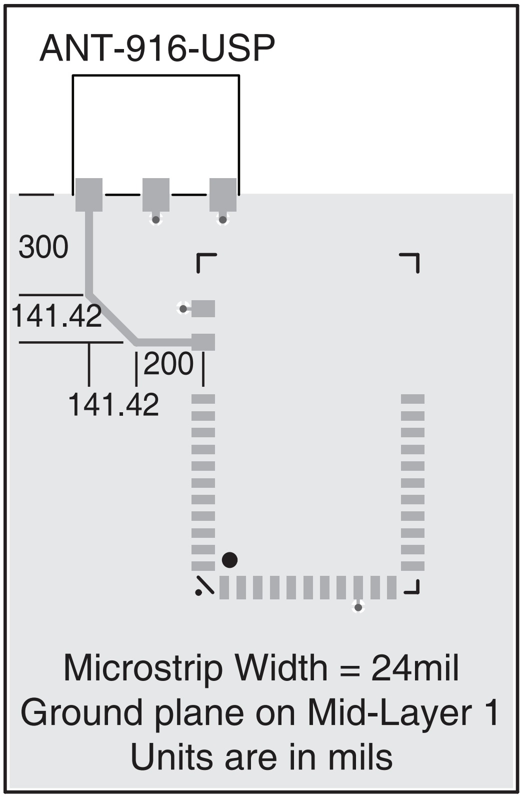


Figure 5: HUM-A-900-CAS with the MicroSplatch Antenna PCB Layout

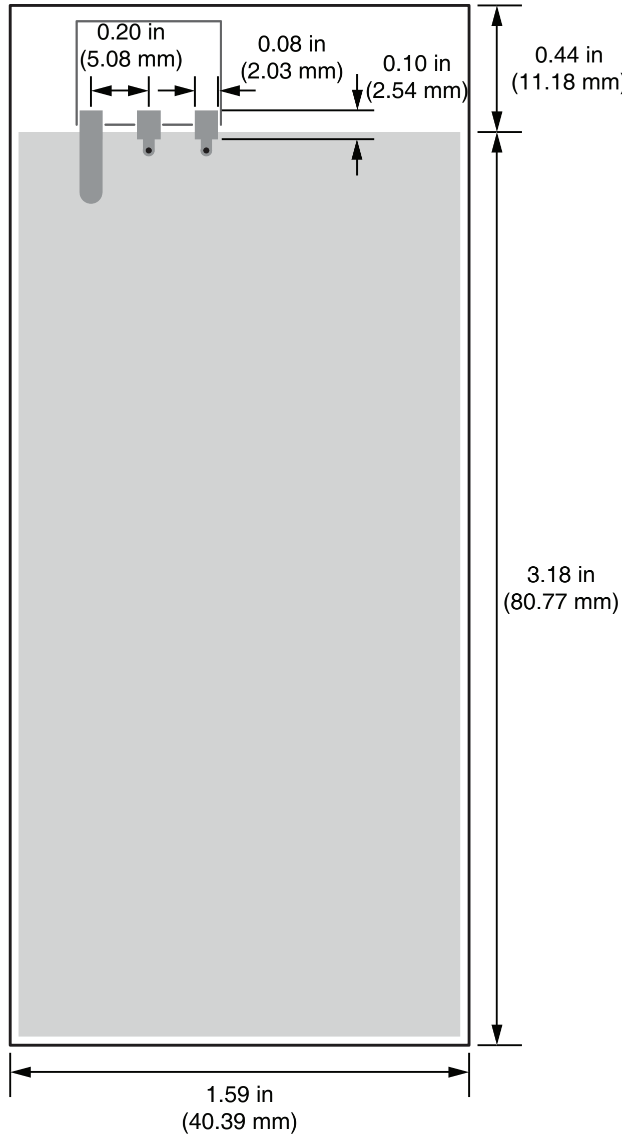


Figure 6: ANT-900-USP Test Fixture PCB Layout

Vertical Mount RP-SMA connection

This RF connection is approved only for operation at the low RF data rate (19.2kbps). This design uses a PCB microstrip to connect the module's antenna castellation to a vertical reverse polarity SMA connector (Linx part number CONREVSMA001). A four-layer PCB is used for this design. A microstrip trace is formed by the 14mil wide trace on the top layer and the ground plane on Mid-Layer 1.

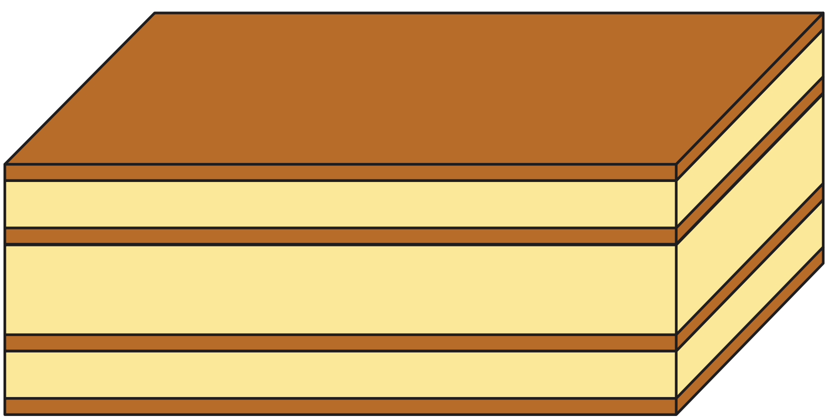
Figure 7 shows the required PCB layer stack. This layer stack up must be matched precisely including material type, dielectric constant, dielectric thickness, and copper thickness.

Figure 8 shows the trace dimensions that must be followed precisely, including trace width, routing, and RF matching components. A 15pF series capacitor (Murata GRM0335C1H150GA01D) is located between the module and the RP-SMA connector. A 10mil wide microstrip connects the center post of the RP-SMA connector to a 100nH inductor (Bourns CW201212-R10J).

The RP-SMA connector is mounted on the bottom of the board, opposite the module.

The Ground plane on Mid-layer 1 must not have any cutouts in the area under the RF trace or the area between the module and the antenna.

Other components required by the application can be placed around the module. The ground plane can be used as the common ground for other circuits on the board. The general board layout guidelines should also be followed.



Layer Name	Thickness	Material
Top Layer	0.7mil	Copper
Dielectric 1	6.7mil	FR-4 (Er = 4.6)
Mid-Layer 1	0.7mil	Copper
Dielectric 2	46.0mil	FR-4 (Er = 4.6)
Mid-Layer 2	0.7mil	Copper
Dielectric 3	6.7mil	FR-4 (Er = 4.6)
Bottom Layer	0.7mil	Copper

Figure 7: HUM-A-900-CAS with a Vertical RP-SMA Connector PCB Layer Stack

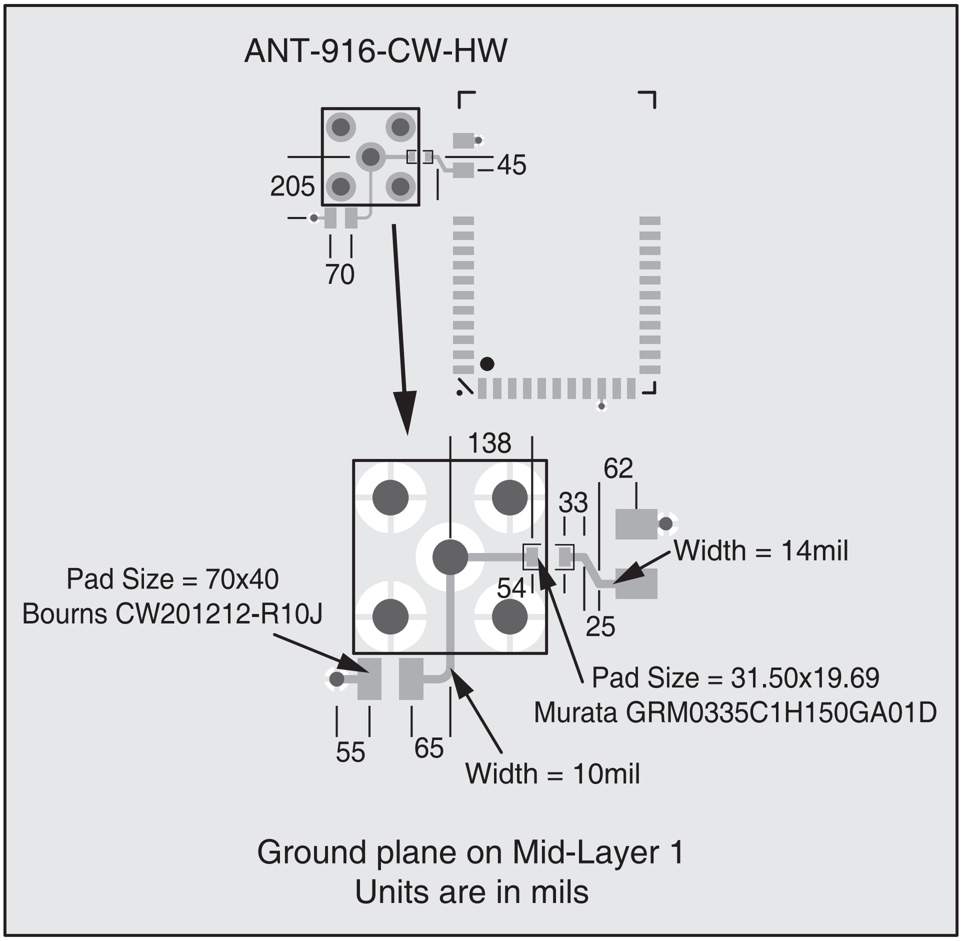


Figure 8: HUM-A-900-CAS with a Vertical RP-SMA Connector PCB Layout

HUM-A-900-PRO-ttt PCB Footprint

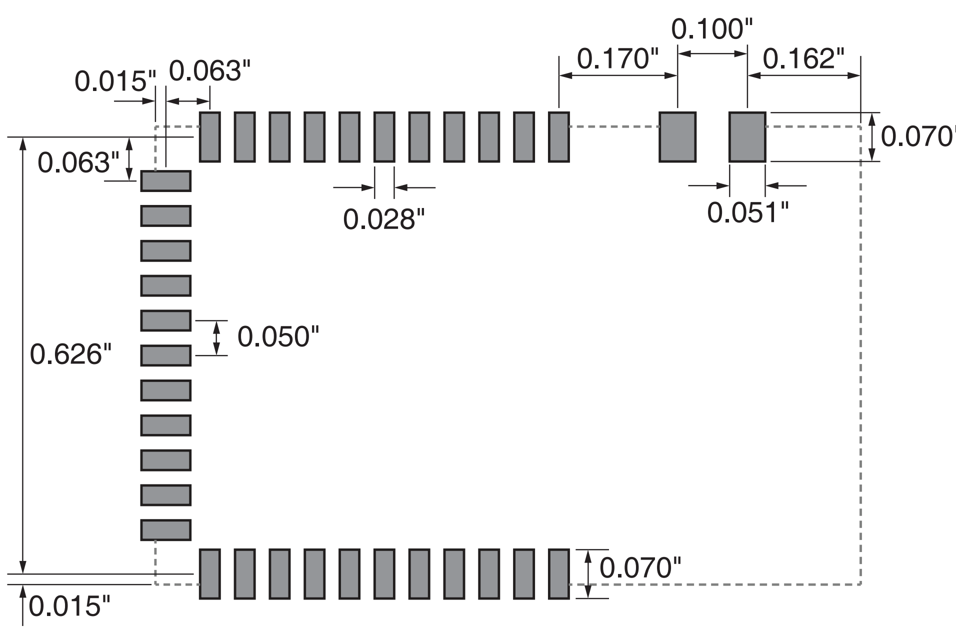


Figure 9: HUM-A-900-ttt PCB Footprint

Approved Antennas

The antennas in Figure 10 are tested and approved for use with the HUM-A-900-PRO Module. 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 (consult specification sheet for cutoff frequencies).*

Contact Linx for information about other antennas that meet these requirements and may be used with the HUM-A-900-PRO module.

Antennas / Antennes				
Linx Part Number Reference Linx	**Type**	**Gain**	**Impedance**	**Valid For**
Tested Antennas				
ANT-916-CW-HWR-RPS	½ Wave Dipole Helical	1.2dBi	50Ω	Both
ANT-916-USP	¼ Wave Planar	0.3dBi	50Ω	–CAS
Antennas of the same type and same or lesser gain				
ANT-916-CW-HW	½ Wave Dipole Helical	1.2dBi	50Ω	Both
Cable Assemblies / Assemblages de Câbles				
Linx Part Number Reference Linx	**Description**			
CSI-RSFB-300-UFFR*	RP-SMA Bulkhead to U.FL with 300mm cable			
CSI-RSFE-300-UFFR*	RP-SMA External Mount Bulkhead to U.FL with 300mm cable			
* Also available in 100mm and 200mm cable length				

Figure 10: Amplified HumPRO™ Series Transceiver Approved Antennas

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 stack up solution used for the build from the PCB manufacturer that specifies dielectric thicknesses and target dielectric constants for substrate materials.

Linx Applications Engineers are available to review Layout designs to ensure compliance and optimal RF performance.

Production Test Procedures for Ensuring Compliance

During production test for the host device, The HUM-A-900-PRO module is to be activated in maximum power transmit mode and the conducted RF output power at the RP-SMA connector 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 output power specified in the HUM-A-900-PRO Data Guide. If a conducted output power test is not possible on the host device, an equivalent radiated output power test may be used.

Board Layout Guidelines

The previous sections provide specific requirements for the implementation of each antenna connection, but there are other general rules that should also be considered. Failure to observe good layout techniques can result in a significant degradation of the module's performance. Grounding, filtering, decoupling, and routing are important considerations for any RF design. The following section provides some general design guidelines.

During prototyping, the module should be soldered to a properly laid-out circuit board. The use of prototyping or "perf" boards results in poor performance and is strongly discouraged. Likewise, the use of sockets can have a negative impact on the performance of the module and is discouraged.

The module should, as much as reasonably possible, be isolated from other components on your PCB, especially high-frequency circuitry such as crystal oscillators, switching power supplies, and high-speed bus lines.

When possible, separate RF and digital circuits into different PCB regions.

Make sure internal wiring is routed away from the module and antenna and is secured to prevent displacement.

Do not route PCB traces directly under the module. There should not be any copper or traces under the module on the same layer as the module, just bare PCB. The underside of the module has traces and vias that could short or couple to traces on the product's circuit board.

Each of the module's ground pins should have short traces tying immediately to the ground plane through a via.

Bypass caps should be low ESR ceramic types and located directly adjacent to the pin they are serving.

In some instances, a designer may wish to encapsulate or "pot" the product. There are a wide variety of potting compounds with varying dielectric properties. Since such compounds can considerably impact RF performance and the ability to rework or service the product, it is the responsibility of the designer to evaluate and qualify the impact and suitability of such materials.

TE TECHNICAL SUPPORT CENTER

USA:	+1 (800) 522-6752
Canada:	+1 (905) 475-6222
Mexico:	+52 (0) 55-1106-0800
Latin/S. America:	+54 (0) 11-4733-2200
Germany:	+49 (0) 6251-133-1999
UK:	+44 (0) 800-267666
France:	+33 (0) 1-3420-8686
Netherlands:	+31 (0) 73-6246-999
China:	+86 (0) 400-820-6015

te.com

08/23 **Original**