How to Successfully Integrate a Wireless Module

By: Dr. Erik Lins

Field Applications Engineering Manager EMEA, Laird Connectivity

“Wireless modules greatly simplify integrating a wireless technology into your product, compared to a bare chipset-based approach. Crucial parts of the RF design are already integrated into the module and an existing module certification can significantly simplify end-product certification testing. However, to take advantage of both the RF design and the certification, it is important to follow certain design recommendations. This white paper will walk through these design recommendations and explain the underlying concepts, leading to a successful module integration.”

Choosing The Right Wireless Technology

Choosing the right wireless technology for your design is always the first step, but this decision is influenced by many constraints like required range, data rate, battery life, security, infrastructure, signal coverage, product size, co-existence, knowledge, time frame, and more. It's beyond the scope of this document to cover this adequately. However, the following resources from the Laird Connectivity Resource Center can provide general background information:

- **5 Factors to Consider if You're Building a Wireless IIoT Solution**
- **Connectivity Choices for your Medical Device and IoMT Application**
- **Why Choose Cellular for Your Wireless Solution?**
- **Combining Bluetooth and Low Power Cellular for IoT: Practical Design and Innovative Use Cases**
- **Cellular IoT Modem vs. Module: Navigating a Crucial Decision for IoT Design Projects**
- **Regulatory Approvals for Radio Technologies**

For this document let's assume the decision has been made already. The advice given in this white paper applies to all wireless technologies when it comes to a module's integration.

Why Care About RF Design if Everything is in the Module?

While it's true that crucial parts of the RF design as well as the antenna are integrated in the module, RF does not abruptly stop at the module's outer edges. Given this, additional parts and components of the system need to be considered part of the whole RF design. Design recommendations take this into account and are indicated in module datasheets.

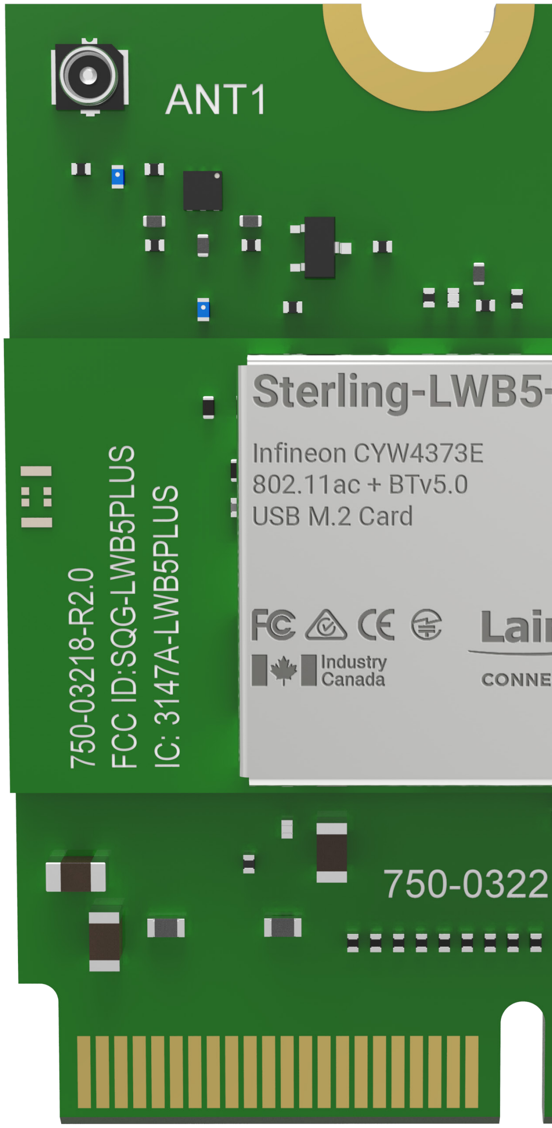
Poor wireless directly correlates with a poor user experience, so make it your earliest PCB inclusion and follow best practices to set your design on a path to success.

General Design Recommendations

The design recommendations in this white paper mainly cover the placement of a module on a circuit board, module placement with respect to ground plane, and routing of crucial signal traces. Recommendations are also made regarding more general issues like power supply or handling of special purpose module signals.

These design recommendations are usually fully implemented on our development kits (DVKs), so these kits (and associated design documents) serve as a reference which are available for most DVKs on their module product pages at www.lairdconnect.com.

In the following sections, we'll browse through several design topics. We will not dive deeply into antenna and RF theory here but will provide some basic information to shed new light onto and motivate the design recommendations from the module datasheets.



Embedded Antennas

Embedded antennas used on wireless modules are usually chip antennas or PCB trace antennas. Both are technically monopole antennas, hence they lack one half of a full dipole. So how can they still radiate properly?

Ground Plane

An RF antenna is usually carried out as half-wave dipole (Figure 1 left). However, it's possible to use just one monopole half of the dipole placed over a ground plane, and by leveraging the physical "method of mirror charges", the second half of the dipole is mirrored into the ground plane (Figure 1 middle).

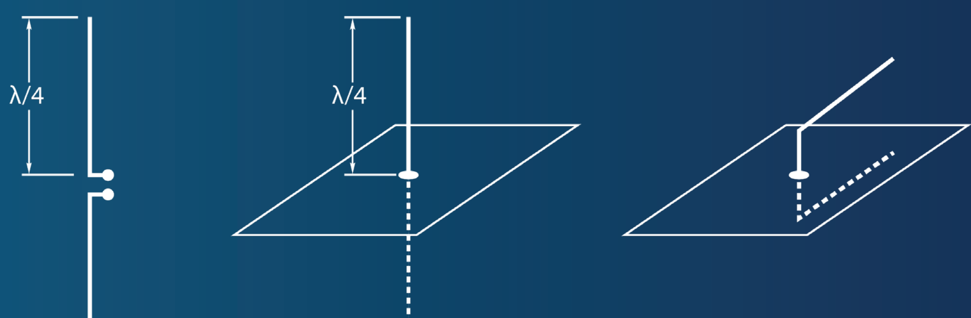


Figure 1 : Antenna evolution from the half-wave dipole (left): quarter-wave monopole over a ground plane (center), L-antenna (right).

This way the monopole antenna plus its mirrored counterpart becomes a full antenna and can radiate efficiently. To reduce antenna size, it's also possible to bend the antenna into an L-shape or even further into an inverted F-shape (IFA) antenna (Figure 2).

From these pictures, we can see that to efficiently leverage the mirrored part of the antenna, the ground plane needs to be both solid and of sufficient size. If the ground plane is too small, the mirror image of the antenna is fully "visible" only at a reduced angular range and this will introduce a more pronounced directivity of the antenna, which can limit the practical range in the final application. The same applies in cases where the ground plane is not solid and has significant openings.

Figure 2: Inverted "F" Antenna evolution from the L-antenna by feeding the antenna at a more favorable impedance point (left): extruded version of Inverted F antenna to produced Planar Inverted F antenna or PIFA (right).

In practice, embedded antennas usually do not stand upright over the ground plane as shown in Figure 1 but are arranged in the same plane either as chip antenna or as PCB trace antenna. Given this, the above three-dimensional approach with a perpendicular ground plane for mirroring the antenna must be applied to a two-dimensional setup. The top image of Figure 3 shows a setup with a ground plane arranged in-plane but still "underneath" the antenna.

To illustrate that the mirrored part of the antenna is effectively contributing to the overall radiation, the right image of Figure 3 shows a simulation of a monopole IFA antenna with its ground plane.

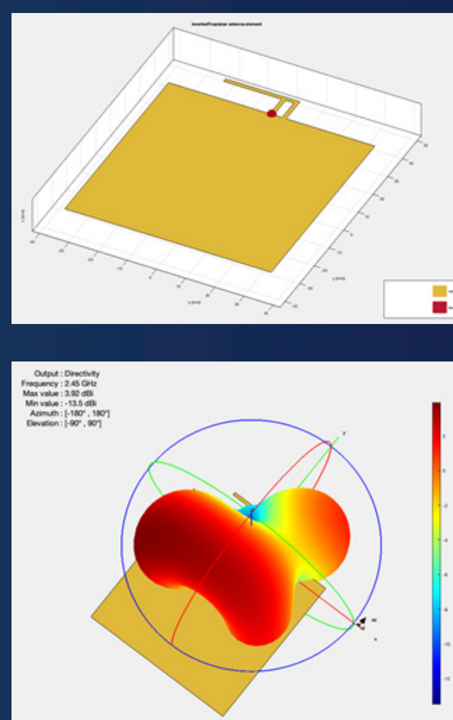


Figure 3: IFA antenna with ground plane (top and simulated radiation pattern (bottom).

We can clearly see that not only the monopole part of the antenna radiates, but so does the mirrored region in the ground plane. If the ground plane were to be too small, part of the radiation lobes would no longer fit into the area of the ground plane, reducing efficiency and range.

The minimum required size of the ground plane depends mainly on the operating frequency of the wireless system. A rule of thumb here would be a size of roughly a quarter wavelength in either direction, like ~30mm for 2.4GHz (Bluetooth or Wi-Fi) or ~80mm for 868/915MHz (LoRaWAN).

However, depending on the requirements of the final application, a more compact overall size might be favorable at the cost of a reduced range. A wearable device communicating over Bluetooth to a smartphone, which is usually nearby, does not require long range, but would be less convenient if it were to have a more obtrusive form factor.

Antenna Keep-Out Region

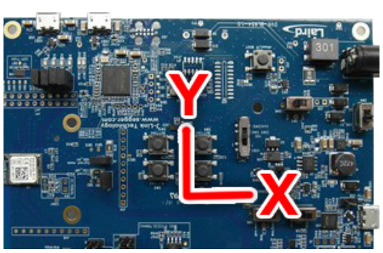
When using a wireless module with an integrated embedded antenna, the ground plane of the circuit board is now separated from the module's antenna. However, the above rules still apply to the combined RF system of circuit board and module. The ground plane is still required for mirroring the antenna, so there is still a need to extend the module area up to where the antenna section starts. Likewise, designers must take care not to protrude other circuitry into the antenna region to avoid shielding/influencing of the antenna. To account for this, a module's data sheet defines a ground plane area with a certain keep-out region for the antenna. We will explore this later in further detail.

PCB Cut-Outs

Since the ground plane belongs to the overall antenna system and contributes to radiation significantly, it's important to avoid large holes or cut-outs in the circuit board. These can behave like slot-type antennas and part of the RF energy can stimulate them to radiate at undesired frequencies. In turn, this might introduce unwanted RF peaks when it comes to EMC compliance testing.

MODULE-SPECIFIC DESIGN RECOMMENDATIONS

Let's have a closer look at a real-world example: actual design recommendations from a BL654 module (451-00001). The datasheet is available at https://www.lairdconnect.com/documentation/datasheet-bl654 and contains useful information spread across various content sections. Not all of them are labeled "recommendations", but it's best to try to follow most noted design practices or at least check them carefully if applicable for your respective design.



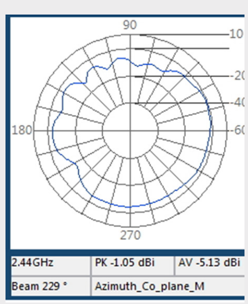
451-00001 On-board PCB Antenna Characteristics

The 451-00001 on-board PCB trace monopole antenna radiated performance depends on the host PCB layout.

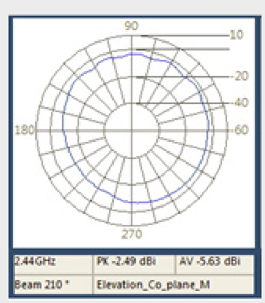
The BL654 development board was used for BL654 development and the 451-00001 PCB antenna performance evaluation. To obtain similar performance, follow guidelines in section PCB Layout on Host PCB for the 451-00001 to allow the on-board PCB antenna to radiate and reduce proximity effects due to nearby host PCB GND copper or metal covers.

Unit in dBi @2440MHz	XY-plane		XZ-plane		YZ-plane	
	Peak	Avg	Peak	Avg	Peak	Avg
451-00001 PCB trace antenna	-1.05	-5.13	-2.49	-5.63	-1.51	-7.43

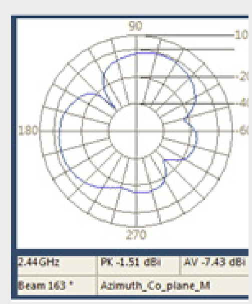
XY Plane



XZ Plane



YZ Plane



Antenna Characteristics

This section presents antenna parameters like overall gain as well as the radiation pattern. Since we established that it's always about the combined system of module and antenna plus the circuit board with ground plane, the BL654 development kit (DVK) was chosen as a reference circuit board. All antenna parameters are based on this design and can vary according to the actual circuit board and setup, hence it's good practice to not deviate from this design too much, especially for the module area.

Hardware Integration Suggestions

Even though thise section is called "suggestions", it addresses issues which are mandatory to follow or at least would need to be carefully checked if they apply to your respective design.

Schematic Design

The first part here is related to the schematic design and provides useful information on various subsystems of the module, such as the power supply system, peripherals, or use of special purpose signals. Figure 5 shows part of these for the BL654 and the actual list in the data sheet is much longer. It's good practice to go through all these points before/during/after the schematic design and carefully check if they have been implemented or at least made sure they are not applicable in your case.

Circuit

The BL654 is easy to integrate, requiring no external components on your board apart from those which you require for development and in your end application. The following are suggestions for your design for the best performance and functionality.

Checklist for BL654 power supply options:

Option 1 – Normal voltage power supply mode entered when the external supply voltage is connected to both the VDD and VDDH pins (so that VDD equals VDD_HV). Connect external supply within range 1.7V to 3.6V range to BL654 VDD and VDD_HV pins.

OR

Option 2 – High voltage mode power supply mode (using BL654 VDD_HV pin) entered when the external supply voltage in ONLY connected to the VDDH pin and the VDD pin is not connected to any external voltage supply. Connect external supply within range 2.5V to 5.5V range to BL654 VDD_HV pin. BL654 VDD pin left unconnected. Nordic Errata 197 and 202 related to the use of VDD_HV DCDC convertor, for details refer to http://infocenter.nordicsemi.com/pdf/nRF52840_Rev_1_Errata_v1.1.pdf. Nordic Errata 202 means no external current draw (from VDD pin) is allowed during power up and VDD_HV rise time (to 3V) is below 1mS.

For either option, if you use USB interface then the BL654 VBUS pin must be connected to external supply within the range 4.35V to 5.5V. When using the BL654 VBUS pin, you MUST externally fit a 4.7uF to ground.

External power source should be within the operating range, rise time and noise/ripple specification of the BL654. Add decoupling capacitors for filtering the external source. Power-on reset circuitry within BL654 series module incorporates brown-out detector, thus simplifying your power supply design. Upon application of power, the internal power-on reset ensures that the module starts correctly

Figure 5: Hardware integration suggestions for schematic design.

Usually, the most crucial points usually are the power supply system as well as an external antenna connection e.g., when using an additional NFC antenna for the BL654 or when using a module with an RF trace pin for an own external antenna. Additionally, signals to external peripherals like e.g., sensors should be checked. They might operate at different voltage levels, hence require some kind of level-shifting. It's also worth checking the actual direction of signals to avoid any conflicts here. E.g., UART TX and RX signals are well known for misinterpreting direction labels sometimes.

Layout Design

Now we get closer to the RF and antenna design issues we addressed in the upper sections of this document. A proper PCB layout is key to a well performing overall module-based design. The below picture from the BL654 data sheet shows a PCB layout checklist. The issues here are important to implement and will ensure good RF performance and are also key to avoid EMC issues when it comes to lab testing and certification measurements on the end-product.

If it seems that GND is an outsized part of these recommendations, see again section Ground Plane information, page 3 on how embedded monopole antennas work and the importance of keeping a ground plane of sufficient size and as solid as possible over as many layers as possible. Also important, and noted here, is module position respective to the circuit board outline.

PCB Layout on Host PCB - General

Checklist (for PCB):

- MUST locate BL654 module close to the edge of PCB (mandatory for the 451-00001 for on-board PCB trace antenna to radiate properly).
- Use solid GND plane on inner layer (for best EMC and RF performance).
- All module GND pins MUST be connected to host PCB GND.
- Place GND vias close to module GND pads as possible.
- Unused PCB area on surface layer can flooded with copper but place GND vias regularly to connect the copper flood to the inner GND plane. If GND flood copper is on the bottom of the module, then connect it with GND vias to the inner GND plane.
- Route traces to avoid noise being picked up on VDD, VDDH, VBUS supply and AIN (analogue) and SIO (digital) traces.
- Ensure no exposed copper is on the underside of the module (refer to land pattern of BL654 development board).

Figure 6: Hardware integration suggestions for PCB layout design.

We also see a mentioning of the PCB layer stack-up here. It's good practice to use a solid ground plane on an inner PCB layer, preferably the one directly underneath the top layer. This layer should be solid ground with no other signal traces being routed there. Unused areas on other layers should be flooded with copper and connected to ground and to the solid inner ground layer by a sufficient amount of GND vias. A known best-practice 4-layer PCB stack-up would be as follows:

Top Layer	Wireless module and other components	Only short tracks to other components. For longer distances, go to Inner Layer 2 and route there.
Inner Layer 1 Avoid any tracks or	Solid ground plane	Avoid any tracks or non-ground areas here. Inner Layer 2 Signal
Inner Layer 2 Signal layer	Signal layer	Tracks to more distant components.
Bottom Layer	2nd ground plane	Alternative signal layer if needed.

Note: A given antenna keep-out region (see below) applies to ALL layers, so no copper/components here on any layer!

Whilst a 4-layer circuit board usually gives best results in terms of RF and EMC performance, it's also possible to achieve reasonable performance with a simpler, lower-cost 2-layer board. However, the above layout recommendations need to be followed as closely as possible within those two layers

Antenna Ground Plane Keep-Out

While the above layout recommendations address more general issues and apply to any RF design, this section gives additional recommendations for a given module. See Figure 7 for this section from the BL654 data sheet.

The importance of a ground plane and antenna keep-out region has already been established. In Figure 7 we also see a quite significant amount of GND vias (green dots) spread over the unused circuit board areas and filled with ground. This ensures non-floating copper islands and makes the ground plane even more solid regarding RF current flow.

For a convenient adoption of the ground plane and keep-out structure, the data sheet further below contains a detailed mechanical drawing of the module's land pattern as well as keep-out area and module position to the PCB edge, see Figure 8.

Antenna Keep-out on Host PCB

The 451-00001 has an integrated PCB trace antenna and its performance is sensitive to host PCB. It is critical to locate the 451-00001 on the edge of the host PCB (or corner) to allow the antenna to radiate properly. Refer to guidelines in section PCB land pattern and antenna keep-out area for the 451-00001. Some of those guidelines repeated below.

- Ensure there is no copper in the antenna keep-out area on any layers of the host PCB. Keep all mounting hardware and metal clear of the area to allow proper antenna radiation.
- For best antenna performance, place the 451-00001 module on the edge of the host PCB, preferably in the edge center.
- The BL654 development board has the 451-00001 module on the edge of the board (not in the corner). The antenna keep-out area is defined by the BL654 development board which was used for module development and antenna performance evaluation is shown in Figure 10, where the antenna keep-out area is ~5 mm wide, ~39.95 mm long; with PCB dielectric (no copper) height ~1 mm sitting under the 451-00001 PCB trace antenna.
- The 451-00001 PCB trace antenna is tuned when the 451-00001 is sitting on development board (host PCB) with size of 125 mm x 85 mm x 1mm.
- A different host PCB thickness dielectric will have small effect on antenna.
- The antenna-keep-out defined in the Host PCB Land Pattern and Antenna Keep-out for the 451-00001 section.
- Host PCB land pattern and antenna keep-out for the BL654 applies when the 451-00001 is placed in the edge of the host PCB preferably in the edge center. Figure 10 shows an example.

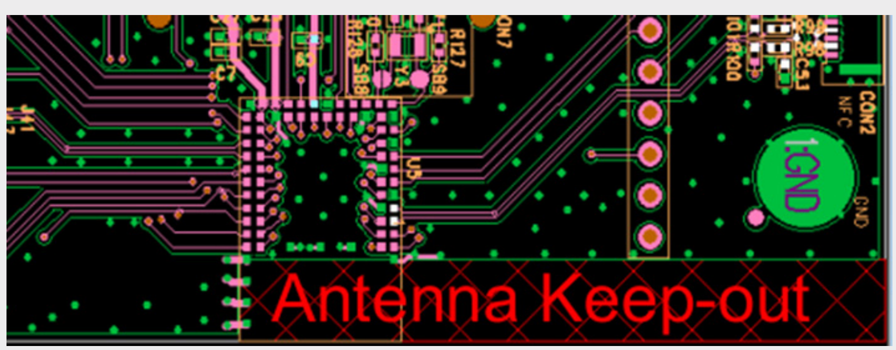


Figure 7: PCB trace antenna keep-out area at PCB edge for the BL654 module.

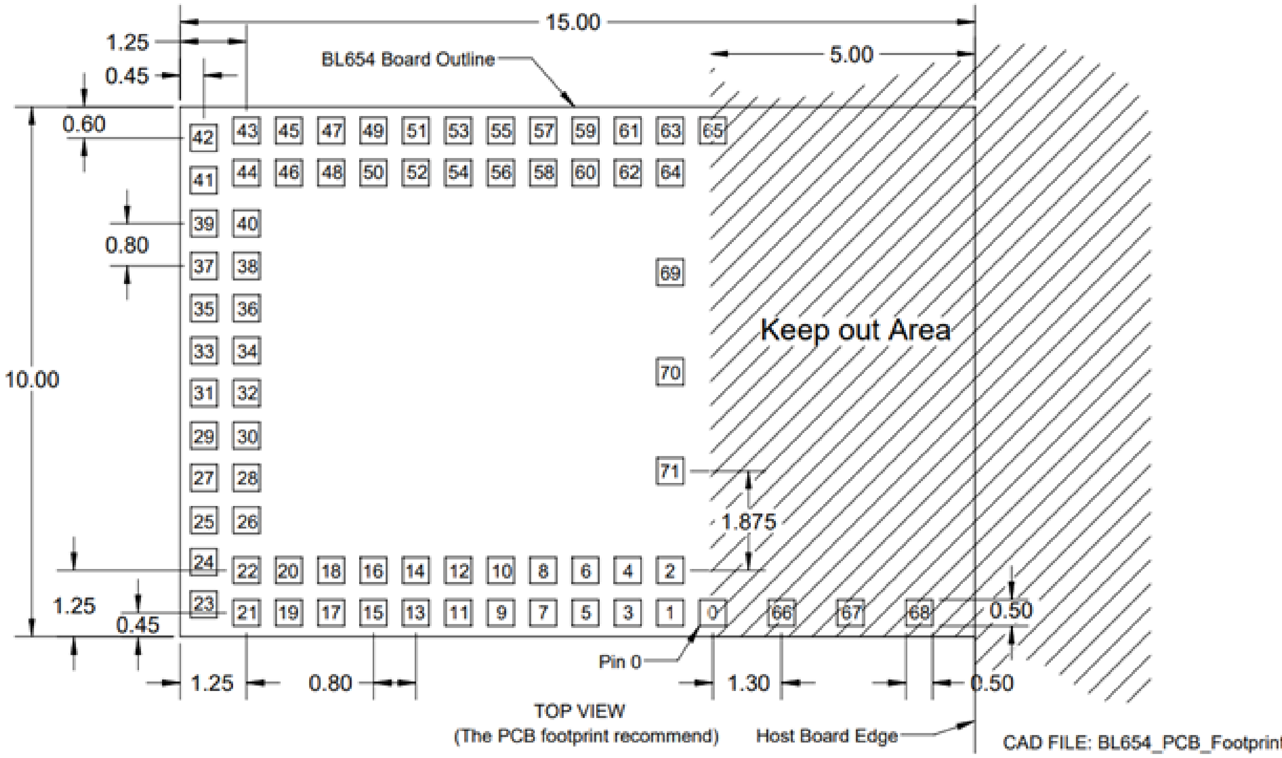
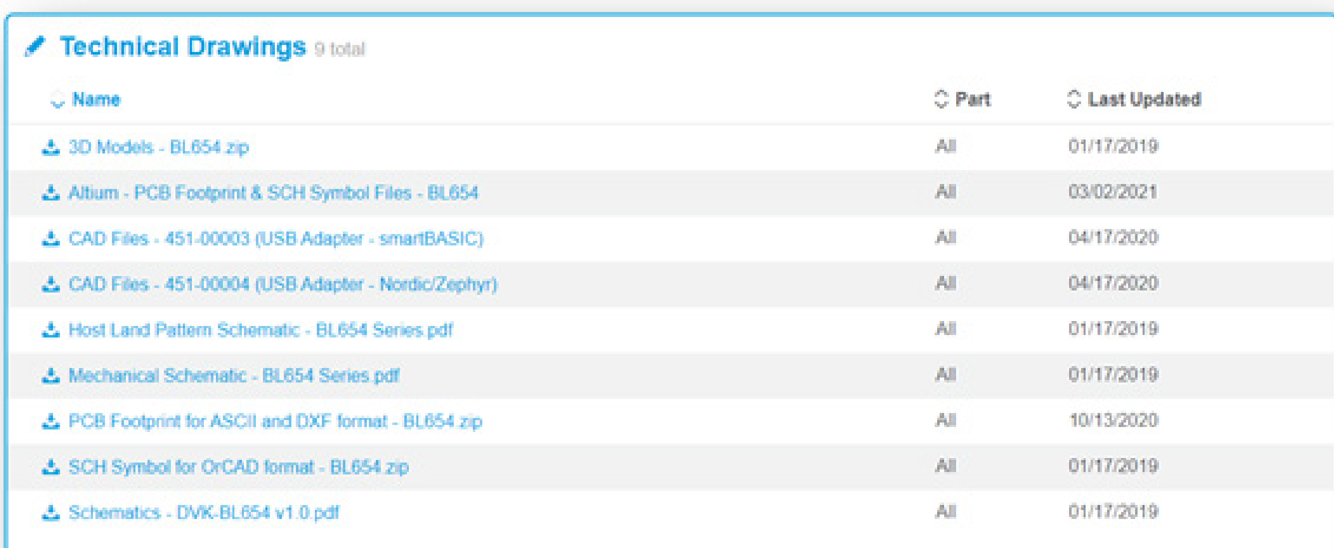


Figure 8: Land pattern and keep-out for the BL654.

Laird Connectivity also provides a library of support materials such as PCB footprints, SCH symbols, 3D models, and schematics of the DVKs. These are found in the Technical Drawings heading of the documentation tab on the product page. Figure 9 shows an example for the BL654 at www.lairdconnect.com/bl654.


Technical Drawings 9 total

Name	Part	Last Updated
3D Models - BL654.zip	All	01/17/2019
Altium - PCB Footprint & SCH Symbol Files - BL654	All	03/02/2021
CAD Files - 451-00003 (USB Adapter - smartBASIC)	All	04/17/2020
CAD Files - 451-00004 (USB Adapter - Nordic/Zephyr)	All	04/17/2020
Host Land Pattern Schematic - BL654 Series.pdf	All	01/17/2019
Mechanical Schematic - BL654 Series.pdf	All	01/17/2019
PCB Footprint for ASCII and DXF format - BL654.zip	All	10/13/2020
SCH Symbol for OrCAD format - BL654.zip	All	01/17/2019
Schematics - DVK-BL654 v1.0.pdf	All	01/17/2019

Figure 9: Technical drawings for the BL654

Further Recommendations

In addition to the above, the physical proximity of any parts to the module and antenna will have performance impacts on the wireless design. Objects, especially metallic objects near the module, will detune the antenna and degrade signal, so you should maintain a minimal distance between the module and other board-level components and enclosure surfaces. This applies to things like batteries, PCB mounting screws, large connectors (e.g., USB charging connector), and more. See Figure 10 for exemplary recommendations from the BL654 data sheet.

Antenna Keep-out and Proximity to Metal or Plastic

Checklist (for metal /plastic enclosure):

- Minimum safe distance for metals without seriously compromising the antenna (tuning) is 40 mm top/bottom and 30 mm left or right.
- Metal close to the 451-00001 PCB trace monopole antenna (bottom, top, left, right, any direction) will have degradation on the antenna performance. The amount of that degradation is entirely system dependent, meaning you will need to perform some testing with your host application.
- Any metal closer than 20 mm will begin to significantly degrade performance (S11, gain, radiation efficiency).
- It is best that you test the range with a mock-up (or actual prototype) of the product to assess effects of enclosure height (and materials, whether metal or plastic).

Figure 10: Recommendations for metal and plastic enclosures

WHAT IF I USE AN EXTERNAL ANTENNA?

While this document focuses on the integration of wireless modules with an on-board antenna, several of Laird Connectivity's wireless modules are also available as a variant with an external antenna connector. Some provide an integrated UFL/MHF4 connector for connecting, for example, an internal flex-type antenna placed inside the enclosure, or an external antenna connected by an SMA pigtail cable. Some modules do not have a UFL/MHF4 connector, but instead provide an RF antenna trace pin to connect to either a UFL/MHF4 connector or an embedded chip antenna on the host PCB.

External or embedded antennas have different design recommendations. Some of the previously discussed considerations become less important, while new considerations arise from the specific antenna's data sheet or even from a certification standpoint.

Embedded Antennas

When using a chip antenna or similar type of on-board embedded antenna on your host PCB, check the antenna's data sheet for design and layout recommendations. These antennas also require a ground plane to operate properly, and most antennas are optimized to a certain position on the circuit board. Embedded antennas also require proper antenna matching for the frequency band and PCB layout. This requires related measurement equipment like a vector network analyzer (VNA) and quite some knowledge to operate and interpret the given measurement results.

Internal Antennas

Internal antennas could be peel-and-stick flex or PCB type antennas, such as those provided by Laird Connectivity at https://www.lairdconnect.com/rf-antennas/internal-antennas, and may include dipole and monopole antennas. A dipole antenna does not depend on a ground plane to radiate properly, but canis still be influenced by metal and other material in close proximity. The closer the dipole is to metal, the greater the proximity effects altering the radiation pattern. Therefore, it's best to keep metal outside the near-field of the antenna. Monopole antennas require a ground plane to radiate, and efficiency still depends on the size of the ground plane.

External Antennas

External antennas like stubby or rubber-duck type are mounted outside an enclosure and connected by an SMA to UFL/MHF4 pigtail to the module. Those also exist as both dipole and monopole variants. While dipoles do not rely on a ground plane and hence are beneficial on non-metal enclosures, monopole antennas need a metal enclosure to act as ground plane for proper radiation.

Selection of Antenna for Modular Approval

For modules that support one of the above antennas, the existing modular approval is typically associated with a set of pre-tested and certified antennas. Using one of the tested antennas greatly assists with end-product certification. However, it's possible to use different antennas by following some basic guidance principles. In that case, it's important to select an antenna of the same type, with equal or lesser gain, and the same radiating characteristics (in-band & out-of-band) as the certified antenna. Not following this guidance will likely not allow you to leverage the modular approval for the end-product certification.

When in doubt, feel free to contact Laird Connectivity regarding design recommendations or whether a particular antenna is a suitable substitute for a pre-tested and certified antenna.

Global Certification

All Laird Connectivity wireless modules hold several certifications for different regulatory bodies around the world. You can access test reports by opening the Certifications section under Documentation on any product page. Certificates and test reports can be downloaded for each region separately. Figure 11 below shows this for the BL654 module at www.lairdconnect.com/bl654.

✓ **Certification** 15 total

Name	Part	Last Updated
AS/NZS Certifications - BL654 Module	All	12/11/2020
AS/NZS Certifications - BL654-USB	All	12/14/2020
DoC - BL654 Bluetooth USB Adapter	All	08/13/2020
EU Certifications - BL654 Module	All	01/13/2021
EU Certifications - BL654-USB	All	12/14/2020
FCC Certifications - BL654 Dongle	All	12/14/2020
FCC Certifications - BL654 Module	All	03/15/2021
FCC/ISED/EU Modular Approval (C2PC) - 802.15.4 (March 2021)	All	03/16/2021
ISED (Canada) - BL654-USB	All	12/14/2020
ISED (Canada) - Certifications - BL654 Module	All	03/15/2021
ISED ICES-003 Issue 7 Declaration of Compliance	All	05/18/2021
MIC Certifications - BL654 Module	All	12/14/2020
Regulatory Information - BL654	All	09/09/2021
Regulatory Information - BL654-Dongles	All	03/23/2021
RoHS 3 - Bluetooth	All	09/23/2021

Figure 11: Certification documents and regulatory information for the BL654 module.

More comprehensive information is given in the "Regulatory Information" document for each product, which covers the below topics:

- Current regulatory certifications for different regions along with the respective regulatory IDs
- Certified antennas tested with the module
- Documentation and labeling requirements when integrating a module into a host device

Integrating a module into a host product technically constitutes a modification of an approved product, and many countries' regulators will require a minimum level of integration testing with the module in its host application.

- For the FCC/ISED, if the trace layout isn't followed exactly, then none of the existing certifications can be leveraged and the device will need to be fully retested. After the test data has been gathered, and the device is still compliant, you may file a Change of ID for the module, taking ownership of the certification from the manufacturer, and then file a Class II Permissive Change (C2PC) to add the new trace to your grant. This increases testing requirements bymultiple of three or four as opposed to leveraging the approved trace, which will require a new number.
- FCC/ISED identification number that you will manage, and labeling on the module must reflect this new number.
- For the EU, if Radio Equipment Directive (RED) is applicable and CE marking is required for your end-product, you must conduct a risk assessment to determine what can and cannot be leveraged based on your design modifications. Less modifications means less risk. Conducted measurements can often be leveraged from the modular approval, but it's best to consult with your test house early in the design process. We advise you follow a similar approach to what FCC/ISED requires in case regulators determine the available data is insufficient.
- For other regions, the situation might differ and may need to be discussed with local regulatory authorities.

Regarding the above, the takeaway is that the approved product (module) will still require some level of host integration testing, but the pre-existing certification of the module will greatly reduce the scope, complexity, and cost. In practice, this means it's good to incorporate as many design recommendations as possible to ensure regulatory compliance when integrating a module into a host device.

STEP-BY-STEP WALKTHROUGH

Since a wireless module (being an RF device) is a crucial part of the overall system and is influenced heavily by other components and parts, it should be considered at a very early stage of the system design. Wireless performance relies on proper placement and layout of the module, while other components like sensors, connectors, and batteries could be more arbitrarily placed on the circuit board without compromising their functionality. Avoid implementing all the rest first and then wondering where to put the module.

- Select an appropriately-sized circuit board with a large enough ground plane for the given wireless frequency.
- Place the wireless module first according to design recommendations (usually at the longer edge of the PCB and sometimes on the corner)
- Keep large metal components like connectors, batteries, and screws at a reasonable distance from the module and its antenna. It's better to run signals a bit longer (e.g. USB signals which require controlled impedance/micro-stripline and should typically be kept short) rather than to compromise the module/antenna.
- Provide a solid ground plane, preferably on an inner PCB layer. Flood unused PCB areas with ground and add a sufficient amount of via connections to that inner ground plane.
- Implement the recommended antenna keep-out area and keep it free from any copper and other components.
- Avoid large holes or cut-outs in your circuit board, because they can act as additional, unwanted slot antennas and radiate at undesired frequencies.
- Implement a suitable power supply architecture for the respective module and check the available power supply options of the module like internal LDO and/or DCDC

converter settings. Voltage supply ranges of recent modules often allow for direct supply from a battery, which is often best with regards to a low voltage supply impedance and current draw.

- Select an appropriate battery technology for your wireless technology. Check the maximum peak currents during RF transmission, which can be significantly higher than the average current consumption. Bluetooth TX current bursts can briefly peak at 15mA when advertising is used, which is simple to buffer with capacitors, hence a coin cell would be a usable battery. LoRaWAN can reach around 45mA for up to 2two seconds on large spreading factors, which makes a coin cell unsuitable and would call for a more powerful cell like lithium thionyl chloride. If a rechargeable battery is required, keep in mind that the cell voltage during charging often is significantly higher than during normal operation, which introduces additional constraints to the power supply system.
- Check the design recommendations for additional signals like configuration signals, reset, serial interfaces, etc.

CONCLUSION

For better performance, stability, longevity of your product, and a better user experience, design recommendations from manufacturers like Laird Connectivity are your friend. Poor wireless directly correlates with a poor user experience, so make it your earliest PCB inclusion and follow best practices to set your design on a path to success. Furthermore, the regulatory process is a common hurdle and is full of potential pitfalls and delays; follow design recommendations to ensure a safe path to market for your design with as minimal hassle as possible.

As always, the experts at Laird Connectivity provide wireless modules, design expertise, EMC and RF testing, and much more. Our decades of experience in wireless product design make us an ideal, trusted partner to bring your wireless designs to life.

Visit www.lairdconnect.com to learn more.

About Laird Connectivity:

Laird Connectivity simplifies wireless connectivity with market-leading RF modules, internal antennas, IoT devices, and custom wireless solutions. Our products are trusted by companies around the world for their wireless performance and reliability. With best-in-class support and comprehensive product development services, we reduce your risk and improve your time-to-market. When you need unmatched wireless performance to connect your applications with security and confidence, Laird Connectivity delivers –no matter what.

For the latest news or more information, visit:
www.lairdconnect.com