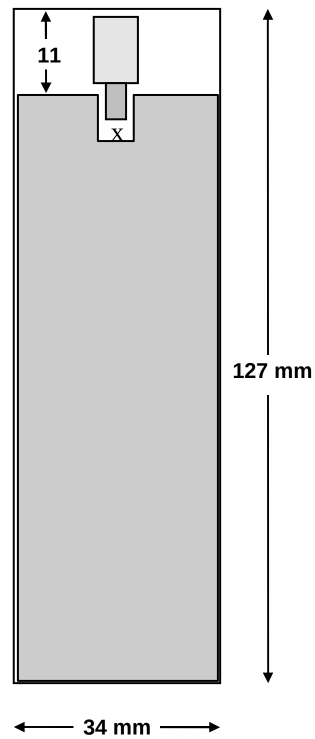
Chip Antenna

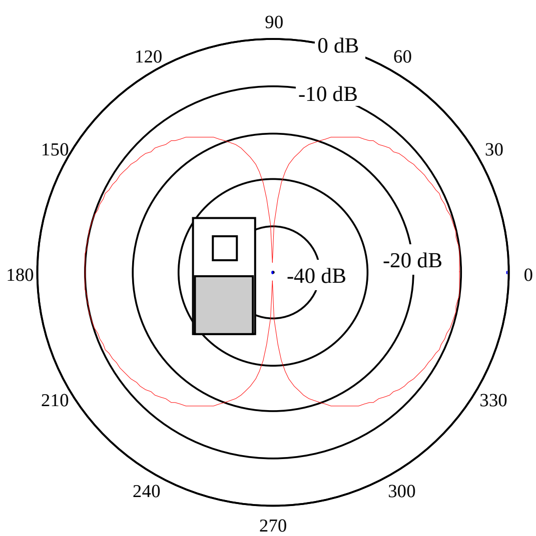
The latest entry into the antenna field is the *chip* antenna. These are surface mount devices that are typically 8 x 5 x 2.5 mm, making them among the smallest design available. They are available for frequencies less than 300 up to 2500 MHz. These antennas are similar to whips in behavior, only much smaller. If an antenna can be reduced in size, while maintaining efficiency, bandwidth will be reduced. So these devices have a very narrow bandwidth and must be made to the exact frequency.

These devices are very groundplane dependant. As a result, they are easily detuned by hand effects, the wrong size groundplane, or the wrong thickness and/or dielectric of the board. The chip antenna must be used according to the manufacturer's recommendations.

For 433.92 MHz, we mounted a chip on a 5 inch long board and obtained a maximum gain of -10 dBd. Not bad when you consider that the spiral has equal gain, but consumes five times as much area on the board. The 916.5 MHz version did better with a 2.6 inch long groundplane for a maximum gain of -3.2 dBd. The polarization is parallel to the long axis of the chip, so maximum radiation is perpendicular to the long axis. There is a deep null, nearly 40 dB, looking at each end of the chip. This would be a big problem if an omnidirectional pattern is required from a horizontal circuit board. When the board is vertical, the pattern is omnidirectional.

Chip Ant, 433.92 MHz





Chip Antenna, 434 MHz

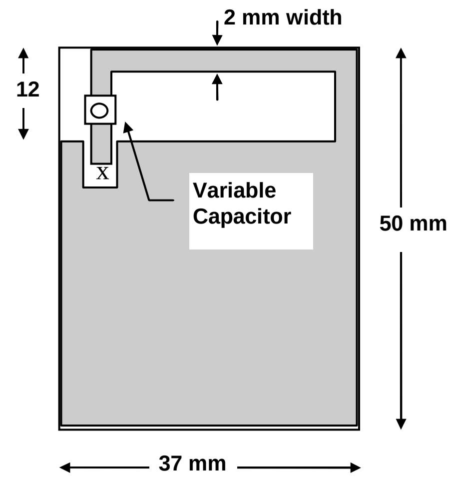
Loop Antenna

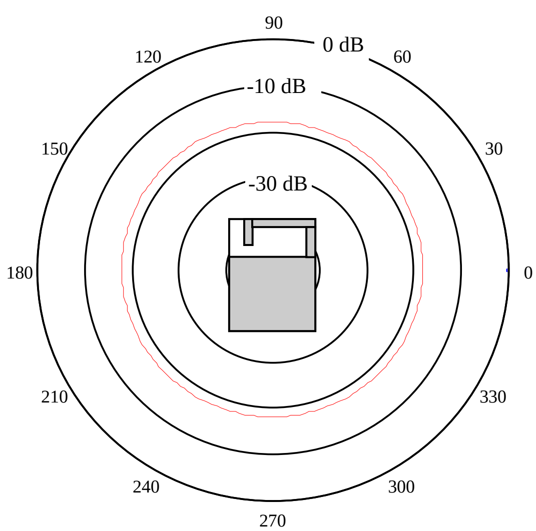
The loop is quite different from a whip, in that both ends of the antenna are terminated. In this case, the end that is opposite the transmitter (or receiver) is grounded. A capacitor is used to tune the antenna to a real impedance, instead of a coil. An advantage of a loop is that it is not easily detuned by hand effects, although the impedance may still vary. The loop can be made small, does not require a groundplane, and takes no more space than a short whip. For these reasons, loops are very common in hand-held devices.

There are some disadvantages. Small loop antennas exhibit relatively poor gain. A small loop will have a very narrow bandwidth. This makes tuning critical. Tuning is often done with a variable capacitor, which adds to the cost, both parts and labor. If the loop is large enough, it may be practical to use a non-variable capacitor. This requires careful adjustment in engineering stages, to ensure that it is properly tuned with a standard value capacitor.

Our example loop antenna covers a 12 x 35 mm area on the end of a board. It is tuned to 433.92 MHz with a variable capacitor. This antenna is very omnidirectional, but had a gain of only -18 dBd. A larger loop should have improved gain.

Loop, 433.9 MHz





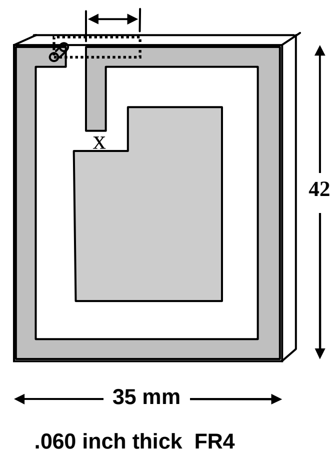
Loop Antenna, 434 MHz

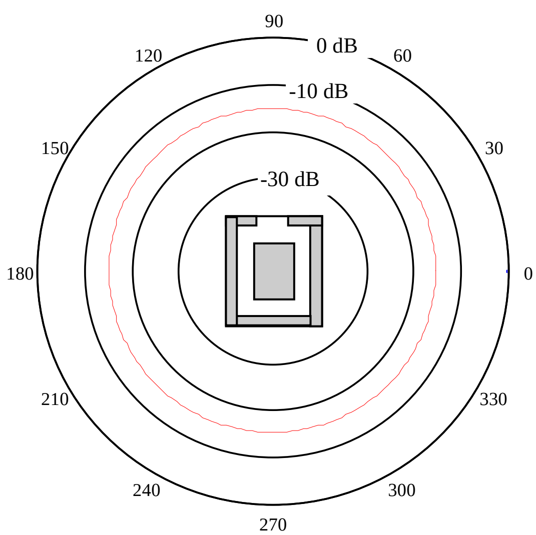
Semi-loop Antenna

This is an unusual design that looks like a loop, but requires no direct grounding. It is comparable to a loop in performance, and can be adjusted to present a non-reactive load. This antenna uses a trace that runs all the way around the edge of a small PCB. The far (open) end is capacitively coupled, through the board, back to the transmitter end of the antenna. The antenna is resonated by varying the length of the short overlapping line. Tuning is not very critical. Hand effects will improve the impedance, with little effect on tuning. Polarization is parallel to the PCB, and the pattern is omnidirectional. Our design had a gain of -15 dBd at 433.92 MHz. This design works very well for hand-held devices.

As with any other designs, this antenna should not run too close to ground. For this design, the transmitter and other circuitry, including battery, should be grouped around the center of the board, leaving the antenna in the clear. The circumference of the board needs to be well under one-quarter wavelength. We have had good results with a circumference of about 0.15 wavelength, and a line width of 1.0 to 1.5 mm, when used in the 400 MHz region. If the design is used on a thinner board, the 5 mm overlap will need to be shortened.

Semi-Loop, 433.9 MHz





**Semi-loop Antenna,
434 MHz**

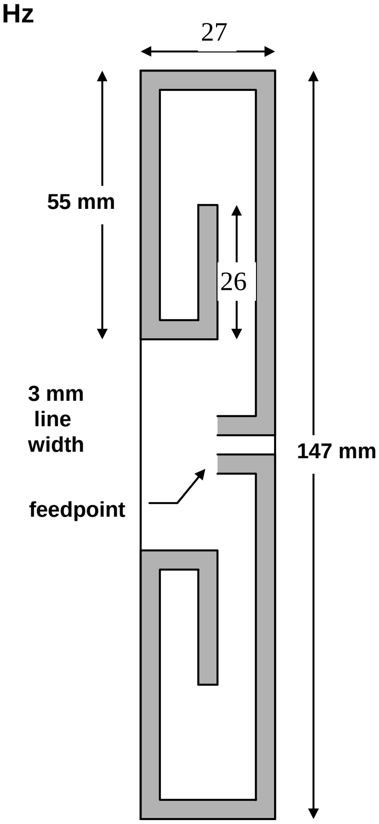
Folded-dipole Antenna

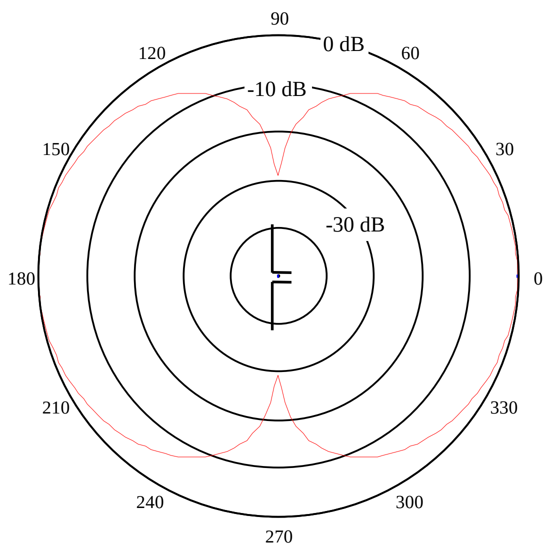
A dipole can be shortened somewhat by bending the wire or line back on itself, but not too close to itself. We built a version on a PCB, shown at right. This antenna has almost the same performance as a full size dipole, but is more compact. The thickness and dielectric constant of the board will affect the tuning, so the length may need to be adjusted.

This type of antenna is an attractive solution where space allows. However, a dipole should not be located close to a large metal area or groundplane. The groundplane will become part of the antenna, and performance will suffer.

Like the normal dipole, the radiation pattern shows deep nulls and good gain. The impedance is a little lower, but still near 50 ohms. Like many of the previous antennas, radiation from the face of the board is just as strong as from the long edge.

Folded Dipole, 433.9 MHz





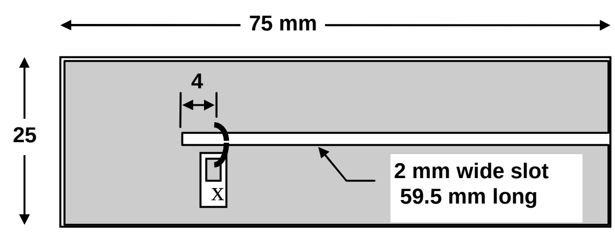
Folded-dipole Antenna, 434 MHz

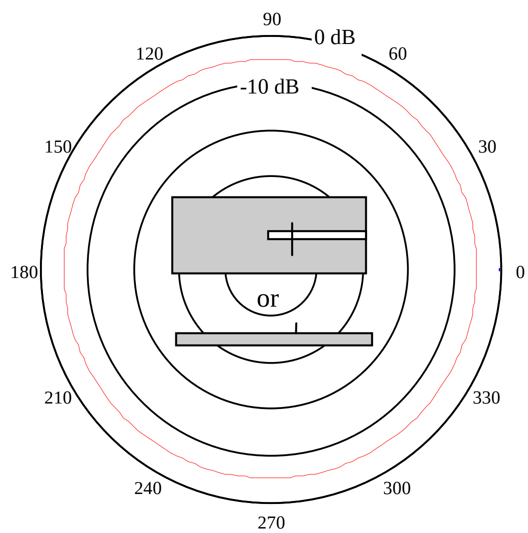
Slot Antenna

Common in radar systems and/or on aircraft, a variation of the slot antenna may have potential above 800 MHz. A quarter-wave slot is cut into a metal sheet or unetched PCB, and if enough area is available, will provide omnidirectional coverage. Our sample antenna at 916.5 MHz required a 75 mm long PCB. The length of the slot was 59.5 mm for 0.060 inch (1.5 mm) thick FR4. A different thickness or dielectric will require changing the length of the slot. One end of the slot must be left open. The slot was fed near the closed end, in this case 4 mm from the end. The feedpoint impedance can be adjusted by moving the feed toward or away from the closed end. Tuning is somewhat critical.

When the board is horizontal, the pattern is omnidirectional around the edge of the board, thus horizontally polarized. We also see omnidirectional coverage when the board is vertical (with the slot horizontal). In this case, polarization is vertical! It may not make sense, but a horizontal slot is equivalent to a vertical whip in this case. Gain is -4.5 to -6 dBd. The feed can be a trace on the backside of the board, with a via used to make connection with the top of the board near the slot.

Open Slot, 916.5 MHz





Open Slot Antenna, 916.5 MHz

Patch Antenna

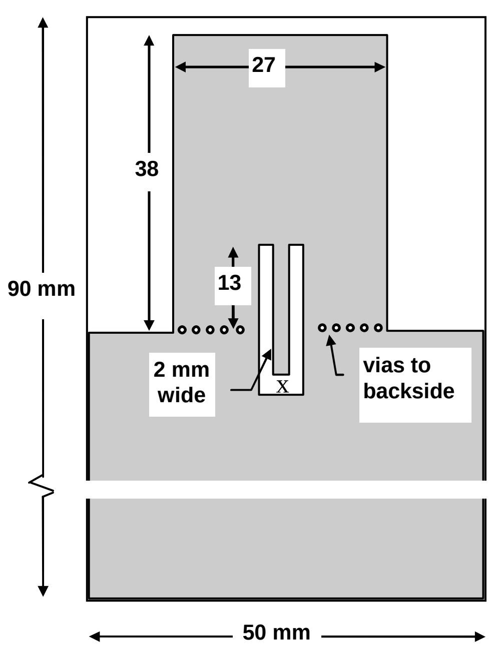
The patch antenna is a low profile design, which consists of a round or rectangular patch of metal very close to a groundplane. The patch is usually printed on a circuit board and can be made as part of the enclosure. Antenna coverage is in any direction above the groundplane, or a hemispherical area. The patch antenna does require a substantial amount of area on a PCB, which makes it more practical above 800 MHz. It has a narrow bandwidth so care must be taken to tune the size of the patch carefully. It is sensitive to the thickness and dielectric constant of the PCB and small variations will mistune the patch completely. It is also sensitive to coatings, but not extremely sensitive to hand effects.

A practical example for 916.5 MHz can fit into an area 30 x 40 mm. The patch size is 27 x 38 mm for a board thickness of 0.060 inch. A thinner board or higher dielectric can require cutting the antenna a little shorter. About 0.1 inch of board space should be left around any ungrounded edge of the patch. One edge of the patch should be grounded with multiple PCB vias. The antenna is fed with a line crossing through the grounded edge to the 50 ohm point on the patch, or by a transmission line coming up through the bottom of the PCB. The 50 ohm point is about 13 mm away from ground on our example patch. The 50 ohm point for any design can be found by moving the vias toward or away from the grounded edge. The farther the feed is away from the ground vias, the higher the impedance will be.

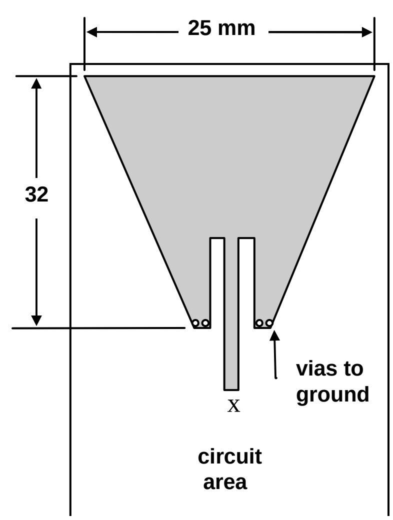
This type of patch is not a full-size, half-wavelength patch, so performance is not as good as a larger size patch. A full-size patch has no grounded edge, so vias are not required. Our example rectangular patch has a gain of -8 dBd. Placing the board against a larger sheet of metal will improve the gain by another 4 dB. If the antenna is made wider than one inch, up to about 3 inches wide, a few more dB can be gained. Polarization is perpendicular to the grounded edge. Gain is good in almost any direction where the patch can be seen, but drops rapidly when looking at the edge of the board.

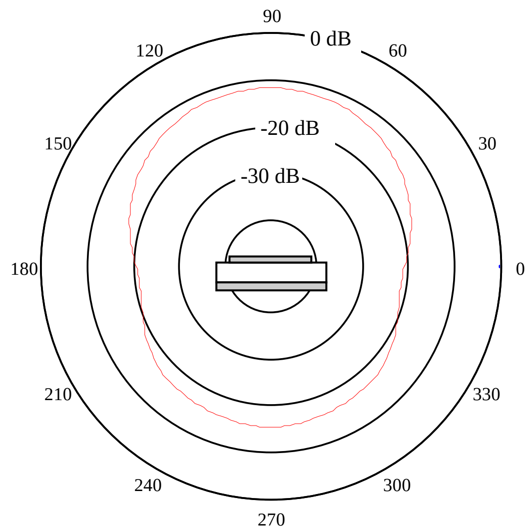
The trapezoidal version has less length so that it can fit into smaller spaces. Patterns and behavior are the same, but the gain is a little lower. We measured about -12 dBd maximum, on a 40 x 90 mm board.

Rectangle Patch, 916.5 MHz

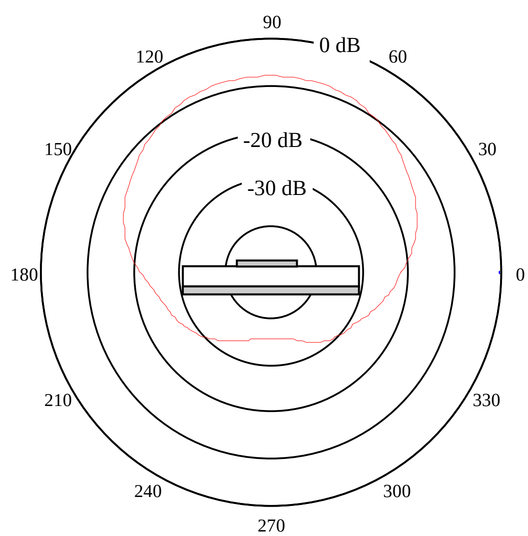


Trapazoidal Patch, 916.5 MHz





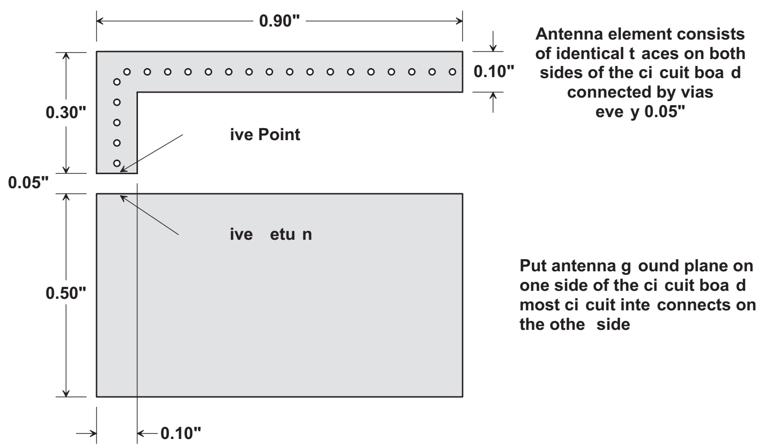
Trapezoidal Patch over a Small Ground Plane, 916.5 MHz



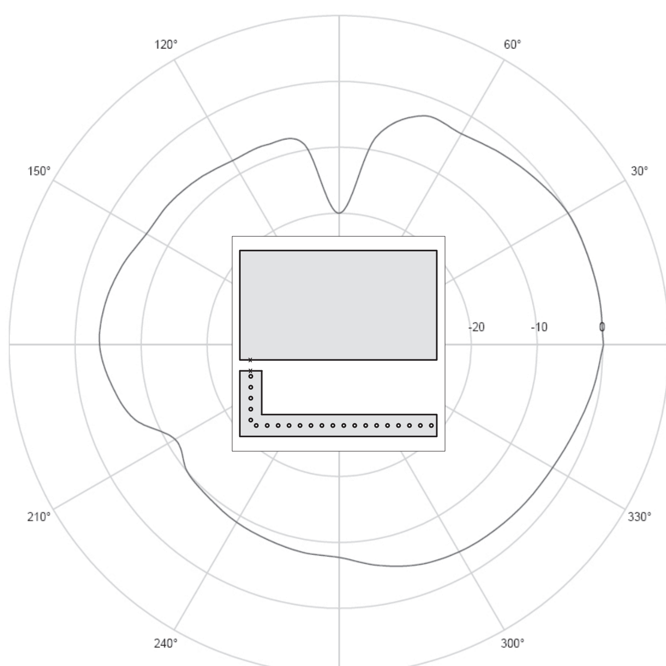
Trapazoidal Patch over a Large Ground Plane, 916.5 MHz

2.4 GHz L Antenna for the TRC104

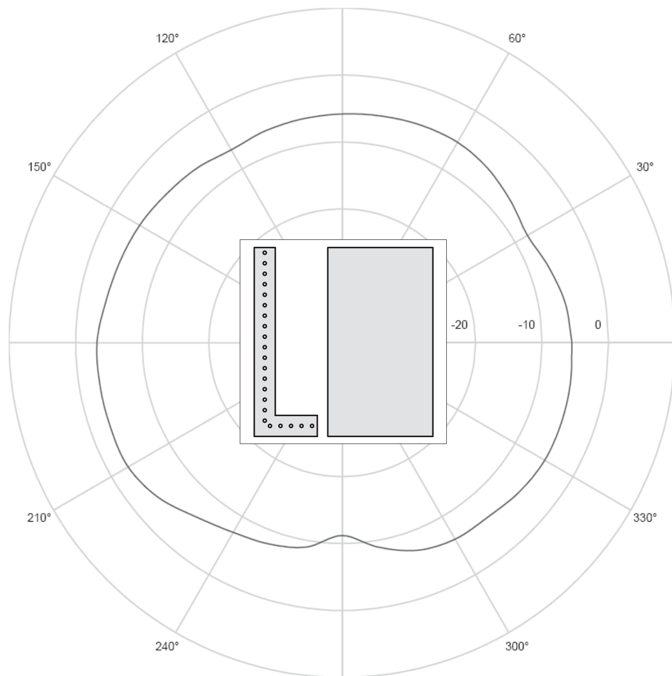
Antennas printed on glass-epoxy PCB material with a dielectric constant of 4.3 to 4.7 can provide good performance for many applications at 2.4 GHz. The L antenna shown below is used on RFM's TRC104 reference design. The antenna exhibits a nominal gain of 0 to -5 dBi depending on the orientation. Note that the driven element consists of identical patterns on the top and the bottom of the circuit board, connected with multiple vias. This makes the antenna relatively insensitive to the thickness and dielectric constant of the circuit board.



L Antenna Dimensions



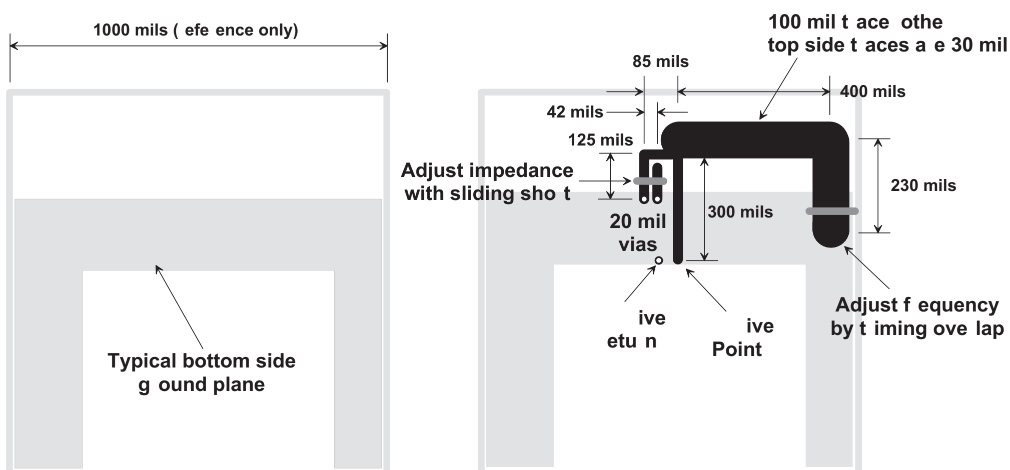
L Antenna Pattern with the Antenna Circuit Board Horizontal and One Meter Above Ground



L Antenna Pattern with the Antenna Circuit Board Vertical and One Meter Above Ground

Tunable 2.4 GHz F Antenna

Another compact circuit board antenna suitable for 2.4 GHz applications is shown below, courtesy of Kent Electronics, www.wa5vjb.com. This antenna can be prototyped and tuned to work with various ground plane configurations and circuit board thicknesses. The resonant frequency of the antenna is adjusted by trimming back the length of the antenna trace on the top side that overlaps the ground plane on the bottom side. A sliding short is used to adjust the impedance of the antenna. After prototyping the antenna with the TRC104 circuitry, the antenna is tuned, and the final antenna dimensions incorporated into the circuit board layout. Tuned antenna performance is similar to the L antenna discussed above.



Use with 32 to 64 mil ci cuit boa d mate ial

F Antenna Dimensions

Enclosures

An antenna should not be located inside a conductive, or metal enclosure. Care should be taken to keep the antenna away from metal surfaces. If a conductive area is large in terms of wavelength (one half wave or more), it can act as a reflector and cause the antenna to not radiate in some directions. If a metal box is used for an enclosure, an external antenna is required.

Testing and Tuning

Antennas seem to be a mystical art. Any change in nearby materials or dimensions can affect antenna performance. Building a published design does not fully guarantee results. Testing an antenna design is necessary, tuning is often required, and there can be pitfalls along the way.

A network analyzer is normally used to test the impedance or VSWR of the antenna. Some antennas that have an impedance near 50 ohms can be tuned by looking at return loss or a VSWR display. Low impedance antennas may require the use of a Smith Chart display to get good results. In this case, the antenna should be tuned to a point near the pure resistance line.

There are other options, such as a spectrum analyzer with a tracking generator, that can be used with a directional coupler. The coupler will feed power to the antenna while feeding the reflected power from the antenna back to the analyzer. The coupler must have an isolation between the Generator and RF Input Port of 20 dB or more. Calibration is done by noting the power readings with a 50 ohm load connected and then unconnected. Using this technique, “return loss” can be measured. If the antenna is near 50 ohms, the return loss back to the RF input port will be high, due to the antenna absorbing most of the power. A good antenna will show as a dip on the screen at the correct frequency. A dip of only 3 or 4 dB (about a 5:1 VSWR) is normal for a low impedance antenna measured on a 50 ohm analyzer. A dip of 9 dB (about 2:1) or more indicates a well-matched antenna in a 50 ohm system. If the dip is not centered at the right frequency, the antenna length or tuning needs to be adjusted.

Antenna measurements of any kind are tricky since the antenna is affected by nearby objects, including the size and shape of the circuit board, and even by the cable connections to the network analyzer. Pass your hand close to the antenna and the dip should move around a little. If it does not, the antenna may not be connected properly. Antennas that are ground plane sensitive may see all additional wires as an extension of that ground. Try wrapping your hand around the cable that goes to the analyzer. If the measurement changes much, you may need to try a different tactic. One way minimize RF currents on the cable is to put several high frequency ferrite toroid beads on the cable or some absorptive material over the cable.

The best way to fine-tune a remote transmitter antenna is by using the transmitter itself. Put an antenna on a spectrum analyzer and try to keep other large metal objects out of the way. Find a place to locate the transmitter that is away from metal and a few feet away from the analyzer. Always locate the transmitter in the exact same spot when testing. If you have a desk that is wood, mark it’s position with a pencil or tape. If hand held, hold it in your hand just above the marking on the desk. Be sure to position your hand, and the rest of your body, the same way during each test. Take a reading of the power level, and tune the antenna to achieve maximum radiated power. The same thing can be done for a receiver. Transmit a signal to it, and adjust the antenna to receive the lowest signal level from the generator.

Common problems with antennas usually involve insufficient free space around the antenna. The antenna cannot run close to ground or any other trace without effecting the antenna performance. This includes traces on the other side of the board, batteries, or any other metal object.

Receiver performance can be degraded by digital circuits. Fast digital switching creates high frequency noise that can cause interference. Keep receiving antennas away from digital circuit traces. Try to keep digital traces short, and run them over a groundplane to help confine the electromagnetic field that is generated by the digital pulses. If an external antenna is used, then use a coaxial cable.

A transmission line for G-10 material that is .060 inch thick requires a trace width of 0.1 inch, half of that for a .030 inch thick board. This results in a 50 ohm transmission line that will carry RF with minimum loss and interference.

High static voltages can damage RF semiconductors and SAW components. We recommend placing an inductor between the antenna and ground to short out any static voltages. For the 400 MHz region, a value near 200 nH is a good choice. At 916.5 MHz, a more appropriate value may be 100 nH. For 2.4 GHz a value of 30 nH can be used. Decoupling coils should not resonate above the transmitter operating frequency.

Acknowledgments

The author would like to thank John Anthes, Harry Boling, and Jeff Koch for their assistance in the preparation of this application note. Additional information was added to this application note in 2009 by Darrell Ash, Patrick Evans, Kent Britain of Kent Electronics, and Frank Perkins.