Minimizing RF PCB electromagnetic emissions

Looking for a way to reduce unwanted signal emission from transceivers? One method is to use appropriate PCB layout techniques, including mechanical shielding.

By Sean Mercer, Ph.D., C.Eng

Most countries regulate the permissible levels of signal conduction and/or radiation from RF equipment. Therefore, it is important to consider electromagnetic emissions and susceptibility issues during the product design cycle. This prevents costly and potentially time-consuming fixes toward the end of a development cycle. To reduce the likelihood of undesirable emissions emanating from your product, many design strategies, including simple modeling, can be used [1, 2].

The use of component shielding and shielded printed circuit board (PCB) traces can avoid unpleasant problems late in a product design cycle. It is wise to mechanically shield noise sensitive circuits to minimize pick-up of unwanted signals. This is typical of synthesizer and voltage controlled oscillator (VCO) circuits where undesired signal pick-up can result in spurs appearing on the synthesizer output. Transceivers with multiple synthesizers for producing two or more local oscillator (LO) signals require excellent mechanical shielding to prevent crosstalk between the synthesizers. Interactions between the different LOs can result in unpredicted spurious signals.

Configuration problems

A PCB is commonly mounted above a metal plate as part of a mechanical housing. This configuration can have problems if unshielded radiating components or traces on the PCB face the metallic housing. One potential problem is the degradation of LO isolation if the LO trace is routed on a PCB layer without any shielding between the trace and the metal housing. Consider the HF transmitter architecture shown in Figure 1. The second intermediate frequency (IF) signal from the preceding circuitry is mixed to the first IF of 45 MHz using the 13 dBm second LO. The 45 MHz first IF signal is then amplified and, using the first LO, mixed down to a transmit signal in the 2–30 MHz frequency range.

It is important to provide adequate rejection of the second LO signal in the IF cascade to ensure that this LO signal does not appear as an unacceptably large spurious signal on the transmitter output. The use of a mixer with good LO-to-RF port rejection is important for this application to minimize the level of the second LO present with the desired first IF signal. The use of a shielded mixer will minimize the radiation of the LO signal that may couple directly from this component into first IF amplifiers. The layout of the ground plane around the mixer interconnects is also critical to maintain the specified LO-to-RF isolation for the mixer. Aside from suppression of wideband noise, one of the functions of the first IF bandpass filter is to reject the second LO signal.

A transceiver design using the architecture outlined in Figure 1 was constructed using a four-layer FR4 PCB as shown in Figure 2a. The second LO synthesizer components and the LO buffer amplifier were placed on the top layer of the PCB and were isolated within a metal shield can. The leaded mixer with a shielded metal case was placed external to the second LO shield can. The second LO signal from the buffer amplifier was routed to the mixer with an approximately 2 cm long microstrip line on the bottom layer of the PCB.

Adequate rejection of the second LO signal (> 70 dB) occurred when the PCB was tested on the bench without

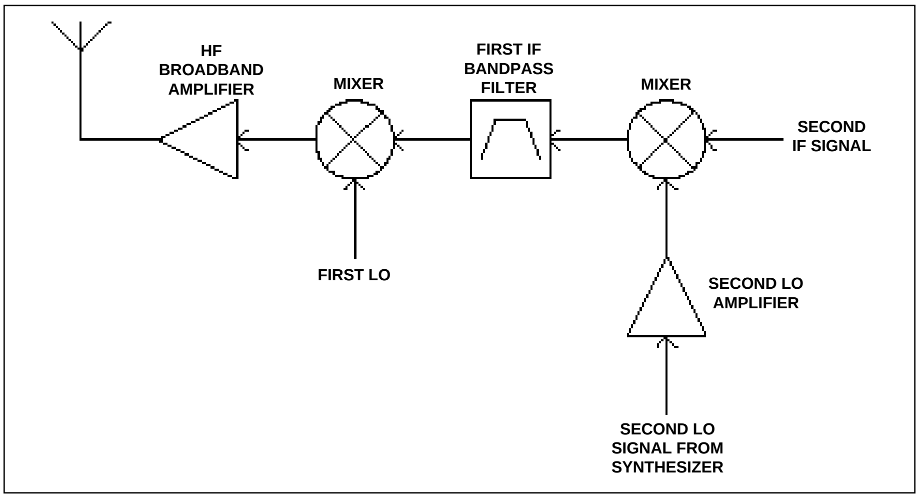

Figure 1. Block diagram of HF transmitter.

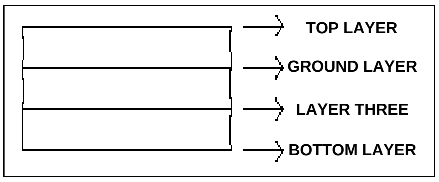

Figure 2a. Four layer PCB.

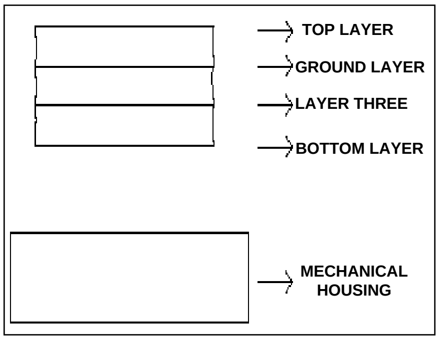

Figure 2b. Four layer PCB with nearby chassis.

its mechanical housing. When the PCB was placed in a metal housing, the bottom layer of the PCB was facing the housing as shown in Figure 2b. The proximity of the metal plate ducted the high-level LO signal past the first IF bandpass filters. The filter rejection of the second LO signal was thereby degraded by some 20 dB. This problem was avoided on subsequent PCB iterations by routing the second LO trace to the mixer as a stripline on layer three of the PCB. The high-level LO signal was then shielded between two groundplanes and no performance degradation was observed when the new PCB was mounted in its mechanical housing.

High-density PCB layouts

Some high-density PCB layouts require component placement on both sides of the board. With components mounted on both sides of the board, it is important to pay attention to both radiating components and traces. Mixers in plastic or ceramic packages will radiate more than a mixer in a grounded metal package. Inductors and active devices such as transistors, field-effect transistors (FETs) and monolithic microwave integrated circuits (MMICs) processing high-level signals can also radiate significant signal levels. If all of the radiating components can be located on one side of the PCB, then shield cans only need to be placed on one side of the board. If radiating components must be located on both sides of the PCB, then shielding may be required on both sides of the board, leading to a more complicated board assembly process. Perforated shield cans that allow increased airflow around the shielded components are commonly used to shield circuitry processing signals under 1 GHz. Note, however, that the third-harmonic of an LO operating around 900 MHz can be in the 2.5–3 GHz frequency range. Many perforated shield cans, depending on the size, spacing and quantity of perforations, will provide degraded isolation at these high frequencies.

A 900 MHz receiver using a high-side injection LO on the front-end mixer to convert the RF input signal to a 45 MHz IF is outlined in Figure 3. The LO signal was routed to the mixer on a microstrip on the bottom of the PCB. The plastic packaged mixer was also mounted on the bottom side of the PCB. The conducted emissions were measured at the receiver input-connector. The levels of LO signal and its harmonics were within specification when the PCB was tested on the bench. The levels of the third-harmonic LO emission degraded some 15 dB when the PCB was mounted into the mechanical housing. The signal radiated from the mixer and the microstrip trace and bypassed the receiver input bandpass filter, coupled into the receiver input and thereby manifested as a spurious signal at the connector. Component shielding and routing the LO signal as a stripline on an inner layer to prevent the signal from radiating off of the PCB were effective measures to reduce the re-radiated signals to within allowable limits. When working at frequencies of hundreds of MHz and beyond, another approach to this problem is to use absorbing material to attenuate the levels of the undesired radiated signals to acceptable levels. This material can be expensive and is probably best used as a last resort to cure stubborn problems rather than a deliberate design strategy.

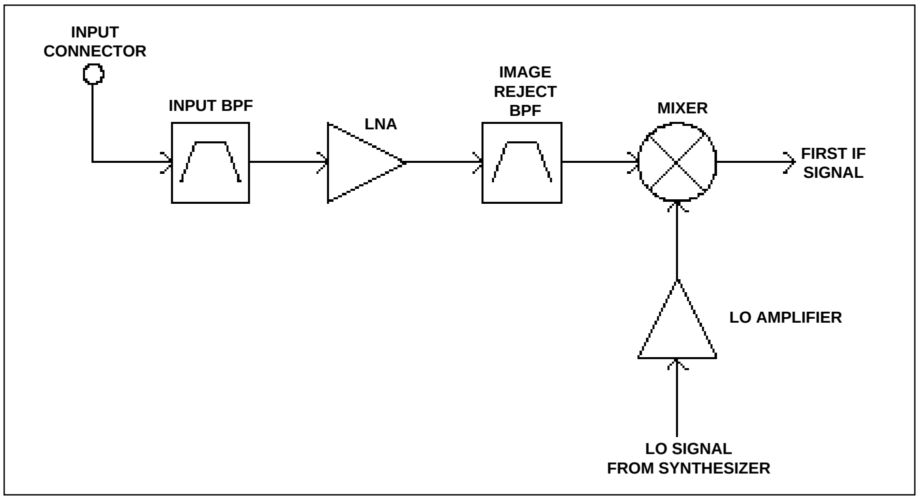

Figure 3. 900 MHz receiver architecture.

Power planes

The use of “power planes” is fairly common on multilayer PCBs with digital circuitry. A separate area or plane of copper is connected to each supply rail. Components are connected to the DC supply by simply connecting to the power plane using a via hole through the PCB. Multiple vias can be used for higher current devices. This approach can, however, be disastrous if used on the RF sections of a PCB. The power planes tend to act as patch antennas and spurious signals can find their way all over the PCB. The safest way to approach DC power distribution on an RF PCB is through the use of adequate width traces to provide the required current capacity and RF decoupling capacitors physically close to each active device.

Mechanical shielding such as a metal shield enclosure can be soldered over a sensitive circuit to reduce susceptibility to radiated spurious signals. This will significantly improve the rejection of radiated signals that may interfere with the operation of a sensitive circuit. Undesired spurious signals can, however, be conducted through the dielectric material on a PCB. The mechanical shield offers no protection against this mechanism of interference. One approach to rejecting this type of signal is to sur-

...continued on page 55

continued from page 48...

round the sensitive circuit with a wall of grounded via holes. If the grounded via holes are placed close enough together, they act in a similar manner to a waveguide below cutoff frequency. The higher the frequency of the signal to be rejected, the closer the vias need to be to each other to provide rejection of that signal [3].

With the integration of RF and digital circuits proceeding at a rapid pace, it is essential for today's RF engineer to be familiar with appropriate interfacing to digital circuitry. The use of programmable synthesizer chips usually requires communication with a microprocessor. Filter capacitors are usually applied to the lines interfacing to the processor. Most synthesizer chips use a three-wire interface consisting of an SPI data bus, the data CLK and the chip enable (CEX) line.

Bus activity causes noise

Bus activity, essentially the toggling of digital lines, is a source of noise and can impact transceiver performance. Keeping the bus inactive during normal transceiver operation can minimize the impact of bus noise on transceiver performance. The high-speed bus is only active when frequency change commands are issued. In situations where bus activity is unavoidable during normal transceiver operation, the most common approach is to use as slow a bus speed as possible to minimize noise and heavily filter the interface lines to the digital circuits that are located on the RF PCB. The data rate over the filtered bus must be slow enough to prevent data corruption caused by the filter attenuation of high frequency components.

If a DC supply line has to be shared between a digital circuit and an RF circuit, then a lowpass filter should be included in the supply line physically close to the RF device. A simple series resistor (small) and shunt capacitor will offer adequate protection for many low-current circuits where the resistor voltage drop can be tolerated and there is little dissipation in the resistor. The voltage drop across the resistor may be intolerable for circuits requiring more than a few tens of milliamps current. A more expensive inductive capacitive (LC) filter may therefore be required for higher-powered circuits.

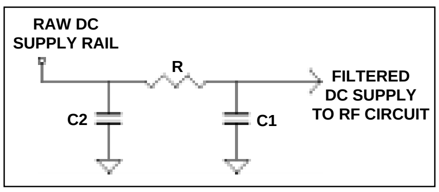

Figure 4. DC supply filter with improved rejection of RF to the raw supply.

Some circuits, such as oscillators or their buffer amplifiers, intentionally generate RF signals in the 0–20 dBm amplitude range. These circuits can introduce RF signals onto the DC supply and distribute them over the PCB. Adding the extra RF decoupling capacitor, C2, to the supply filter shown in Figure 4 can reduce this problem. Active bias filter circuits are commonly used to reject supply-induced noise on sensitive circuits such as LNAs and VCOs.

RF decoupling capacitors

On any PCB where RF signals are present, RF decoupling capacitors should be placed physically close to each integrated circuit or transistor. In the presence of high levels of RF, many devices can exhibit unpredicted behavior such as oscillation. RF decoupling capacitors are a cheap and simple method of avoiding potentially disastrous problems late in a product design cycle. Remember that unnecessary capacitors can be left off of a board as a future cost reduction exercise, but adding a required capacitor will usually require an expensive PCB iteration to provide capacitor mounting-pads, assuming the real estate is available.

There is significant high frequency content in square wave signals with short rise and fall times. As switching speeds in digital circuits increase, impedance matching and line termination become more and more important to minimize signal radiation from PCB traces carrying these signals [4]. The traces between digital signal processing (DSP) chips and external memory devices are a common source of emissions from PCBs. Traces between DSP and memory devices should therefore be kept as short as possible. If a multilayer PCB is used, these traces should also be routed on shielded inner layers to further reduce noise levels.

Conducted emissions from a circuit are often overlooked until the product is tested for regulatory compliance. Fixing the problem at this late stage of a product development can be costly. In some instances, there is no PCB area left to add essential filtering components. In this case, mechanical changes (packaging and PCB outline for example) and an extra PCB iteration will be required. The safest approach to conducted emissions is to plan ahead and include appropriate filtering components on the PCB at the beginning of the design process. If you are fortunate enough to find some of these components unnecessary by the time the product reaches production, it is a simple matter to avoid populating the superfluous parts on the PCB, thereby reducing the component cost for the board.

Mobile applications

Many radio products are required to withstand electrostatic discharge (ESD) to the antenna ports. In an HF radio receiver designed for mobile applications, the receiver may have to withstand a few Watts of RF signal accidentally applied to the receiver. This can occur if there is accidental contact between the antenna's two mobile (vehicle mounted) devices. A diode crowbar connected between the receiver input and ground is an effective protection against this type of accidental input signal. A crowbar configuration consisting of four leaded 1N4150 diodes can be used to protect receivers operating up to 30 MHz without performance degradation. This diode configuration can withstand the continuous application of as much as 8 W of RF without failure. A well-designed receiver protected in this manner should operate without performance degradation once the undesired input signal is removed.

At higher operating frequencies, the protection diodes begin to exhibit significant parasitics, resulting in performance degradation. At frequencies around 900 MHz, the diode crowbar configuration can still be effective if high-speed Schottky diodes are used. A SOT23 packaged Schottky diode pair can cause a front-end noise figure degradation on the order of 0.2 dB, but this is an acceptable trade off for receiver protection in many applications. This type of Schottky diode protection at the low noise amplifier (LNA) can allow a receiver front-end to withstand a continuous RF input level of 20 dBm at 900 MHz without damage at room temperature.

The same caution given above for conducted emission is also applicable to ESD precautions. It is far wiser to include filtering components on the PCB at the beginning of a design cycle than trying to cram them in later. Transorb devices are commonly used to protect digital inputs and outputs from ESD. It is not uncommon to find a Transorb connected at each pin of a D-type connector interfacing to the external environment. PCB layout is also critical for these devices to be effective. Experimenting with a PCB and a static discharge gun will demonstrate the advantage of using Transorb devices. If the discharge applied to a connector pin is not shunted to ground with a Transorb very close to the connector, there is a high risk of the discharge hopping to adjacent traces on the PCB and causing damage to other components. Resistive-capacitive (RC) filtering has been used for ESD protection in some designs. In the application described above, however, RC filtering is almost useless for protecting a circuit from 6 kV contact discharges.

Conclusion

The above examples demonstrate the importance of mechanical shielding for radiating components and PCB traces for minimizing unwanted signal radiation from a PCB. Pay attention to detail when laying out a printed circuit board [5, 6, 7, 8], and consider the board packaging when routing critical traces. Also, remember that high-level signals will cause minimal radiation problems if routed as a stripline trace between two ground planes. To combat this, provide conductive shielding for noise sensitive circuits such as synthesizers and LNAs, and be aware of the frequency range of the signals that the shielding material must attenuate. The isolation of a perforated shielded enclosure will degrade with increasing frequency.

References

1. Vizmuller, P. "Estimating radiated emissions from electronic products," *RF Design*, August 1996, pp. 76-79.

2. Massey, M. "How to design shielding to meet EMI specifications," *RF Design*, November 1996, pp. 54-64.

3. Vizmuller, P. *RF Design Guide, Systems, Circuits and Equations*, Artech House, 1995, pp. 261-264.

4. Breed, G.A. "Notes on High Speed Digital Signals," *RF Design*, Dec. 1994, pp. 70-75.

5. Dye, N., Granberg, H., *Radio Frequency Transistors*, Butterworth-Heinemann, 1993, pp. 92-98.

6. Breed, G.A. "Printed circuit board considerations for low cost design," *RF Design*, September 1994, pp. 62-65.

7. Breed, G.A. "Initial Guidelines for layout of printed circuit boards," *RF Design*, September 1995, pp. 54-58.

8. Trinogga, L.A., Kaizhou, G. and Hunter, I.C., *Practical Microstrip Circuit Design,* Ellis Horwood, 1991, pp. 337-343.

About the author

Sean Mercer is a Senior RF Design Engineer with Glenayre in Vancouver, BC, Canada. He is currently involved in the design of 900 MHz power amplifiers for paging infrastructure applications. He received a M.Sc. (Eng.) in 1987 and a Ph.D. in 1990 from the University of Cape Town, South Africa. Prior to joining Glenayre, Mercer was with Motorola PSG, where this article was written. His past experience ranges from HF transceiver design through amplifier and oscillator designs as high as X band. He can be contacted at 604-293-4399, Ext. 4123 or by e-mail at *sean.mercer@glenayre.com*.