

Key Design Factors for Power and Ground Connections

By Gary Breed
Editorial Director

Power and ground are as much a part of any high frequency circuit as the active and passive components in the signal path

In high frequency engineering, most of the attention goes to the signal path, where we design circuits and systems, analyze their operation and measure the results when completed. What is often overlooked (or over-simplified) is the other connections to those circuits—power and ground. These connections are part of the circuit, and can greatly affect the performance. Because it is not possible to cover this important topic properly in a few pages, this tutorial is intended primarily as a reminder about the most important issues with power and ground connections.

DC Power: Isolate and/or Decouple

When considering the DC power connection, there are two possible objectives. One is isolation, where we want to make the connection disappear, as if there was only a self-contained power source within the device drawing the power. At low frequencies, the usual means of achieving isolation is an inductance (RF choke) that has a high reactance across the operating frequency range. At higher frequencies, a 1/4-wave transmission line may be used to provide the necessary high impedance. When the current draw allows it, a resistor with a value at least five times the system impedance is sometimes used. With any of these techniques, not only must there be sufficient isolation over the operating bandwidth, but the effects outside that range must be predictable—from DC to the maximum frequency supported by the active devices being utilized. Low-frequency oscillations and high frequen-

cy spurious signals are often the result of poor isolation of the power connection.

A circuit's design may require the second option for the DC power connection—effectively grounding it at all pertinent frequencies. Because DC continuity is required, this is accomplished with a bypass capacitor, sometimes in conjunction with additional isolating techniques. The capacitance must be sufficiently high to have a low impedance at the lowest operating frequency, but should not have resonances that create high impedances within the operating bandwidth. Multiple capacitors, or a network of capacitors and inductors (or transmission lines) may be required for wide bandwidth operation.

Problems with DC power are transferred to the active devices they power. Whether low frequency or high frequency, those disturbances will, at a minimum, amplitude modulate the device, adding spurious sidebands to the desired signal. More severe effects are possible if those undesired responses vary bias points or couple directly into the signal path. The potential is great enough that experienced high frequency IC designers almost always include active decoupling to reduce the likelihood of problems. IC applications data typically contains valuable data on how to properly interface the device to the power bus for best reliability.

The primary point regarding DC power connection design is that it is not trivial. Yes, some simple standard practices will achieve acceptable results in most cases, but when performance problems are experienced, it is not unusual for those problems to be traced to the isolating or bypassing components used at the DC power input to the device.

A Checklist for Grounding:

1. There is no such thing as “ground”

This is not a joke. An infinite, perfectly conducting reference plane only exists in theory. In practice, what we call “ground” is no more than an approximation that will always fall short of the ideal.

Even if a perfect ground were possible, when high frequency circuits are constructed in proximity to ground, and the dimensions are significant in terms of wavelength, currents will flow and surface waves will propagate and radiate.

2. The best approximations have low impedance and low loss

Ground planes provide both voltage and signal references, so it is important to make them as unvarying as possible. Large surface area reduces both DC resistance and the concentration of RF currents.

High conductivity metal will minimize the resistance and the currents. Depending on the skin depth at the frequency of operation, copper may be acceptable, but silver or gold plating may be needed for lowest RF loss as the frequency increases.

3. Avoid multiple current paths through ground conductors

If you visualize ground as a rectangular conductive sheet—e.g., one side of a copper-clad printed circuit board—then it is easy to imagine how currents would flow if each corner of that sheet was attached to a larger metal enclosure. DC currents from the center of the sheet will flow to all four corners, varying according to the small changes in resistivity of the copper surface. RF currents at wavelengths in the range of the circuit’s physical dimensions will do the same.

There are two primary means of controlling the flow of currents in this case. First, grounding the board at a single point, rather than at all corners, will eliminate circulating currents (“ground loops”). All cur-

rents flowing from the p.c. board to the enclosure are routed through a single point. This technique is limited to lower RF frequencies where a relatively long path length to ground can be tolerated.

At microwave frequencies, it is common for the circuit ground (the p.c. board in the previous example) to be the same as the enclosure ground. The solid copper backside of substrate materials are firmly pressed to the metal case. To further reduce path length and resistance, the board is often fabricated with multiple via holes, creating an approximation of a solid metal connection between top-side components and the “ground” below.

4. Suspend the reference plane above the larger ground surface

Now we are getting more complex in the way we can deal with the shortcomings of practical grounding. Our “ground” is now a reference plane for the circuitry in the immediate vicinity. Without going into great detail, simply be aware that it is possible to consider a large p.c. board as a series of circuit areas, where we can control both the signal path and the ground behavior.

One of the critical factors in this type of design is controlling the coupling from one circuit block to another, and controlling both ground currents and radiation in the larger enclosure. Localized shielding and isolated “local grounds” are among the common design techniques that enable this type of circuitry. There are even techniques that intentionally treat ground as part of the circuit, inserting discontinuities such as slots for filtering or impedance-transformation purposes.

This method is beyond the scope of a tutorial, but note that it has allowed many microwave systems to be implemented using circuit board assembly instead of individually shielded and isolated modules. It significantly reduces product cost.

5. Design to avoid ground using balanced circuits

At higher microwave frequencies, it is now standard practice to use balanced active circuits and coplanar waveguide transmission lines to reduce the effects of imperfect grounds. In a balanced circuit, ground is virtual, not physical—except at those points where an unbalanced connection is required for interface to an external device.

Balanced circuits increase design and assembly complexity since they have double the number of components. In addition, the construction method must maintain symmetry with the package or enclosure. Coupling between the circuit and its surrounding environment must be same for each complementary side of the circuit.

Also, balanced circuits are more difficult to test. It can be difficult to maintain balance in interconnecting cables. And most test equipment is designed for single-ended connections, which requires calibrated balanced-to-unbalanced transformers with consistent performance.

6. Dealing with ground problems

Circuits at the p.c. board scale are probably the most troublesome, since mass-production techniques may not account for the effects of ground. There are a few EDA tools that can analyze p.c. boards, which is certainly useful, but experienced engineers may not require them. Knowledge of the potential unwanted coupling and RF current paths through “ground” is often sufficient for troubleshooting at the board level.

However, this will not be case with complex multilayer boards being used at higher frequencies. For these structures, as well as for MMIC and microwave modules, EM analysis tools are necessary to identify and avoid problems with grounding.

As a final note, just remember that ground is always part of the circuit you are designing.