Examining The Concept Of Ground In Electromagnetic (EM) Simulation

While circuit simulators require a global ground, EM simulators don't concern themselves with ground at all. As a result, it is the designer's responsibility to ensure that EM results are properly interpreted by the circuit simulator.

BY DR. JOHN DUNN

The notion of ground is one of the fundamental cornerstones of electrical engineering. Defining a ground enables the voltage at a point in a circuit to be described by a single number. However, this seemingly simple concept can lead to confusion when applied to realistic situations. Electrical engineers are familiar with the problem of not having a "good" ground in their circuits. For example, if there is too much resistance or inductance in the ground plane, different points in the ground network can be at different voltages. In this article, we will examine the concept of ground in electromagnetic (EM) simulators.

At first blush, EM simulators don't even address the notion of ground. This raises the question: How do circuit simulators put in a ground so that circuit simulation can be performed? Circuit simulators require an unambiguous, universal ground node, usually referred to as "node 0." This is necessary because simulators solve for voltages, and a unique voltage is required at all nodes.

This article will first introduce the concept of ports in EM simulation, along with how ground is implicitly used when the resulting S-parameters are transferred to the circuit simulator. Next, it will examine the issue of imperfect ground return in the EM simulation and how to properly (and improperly) model it. Three examples will be used to illustrate the concept of ground in EM simulation: a hole in a ground plane, a microstrip to coplanar transition, and a distributed ground return from a package to a board through a ball grid array (BGA) transition.

Ports In EM Simulators And Ground

Simply put, EM simulators solve Maxwell's equations for electric and magnetic fields. Voltage, and therefore ground, is not an EM field concept; rather, it is derived from the electric field by integrating the electric field along a chosen path.[1] If one is working entirely in the world of EM simulation, there is actually no need to discuss ground. It is only when the results are sent to a circuit simulator, usually in the form of an S-parameter file, that ground, or "node 0," must be considered. As a matter of fact, S-parameters can be defined entirely by the concepts of power, modes, and modal impedance, and don't need a voltage definition.[2] However, without such a definition, export to a circuit simulator is impossible.

This article will consider two types of simulators. The first group directly solves for the electric field throughout the entire region of the problem. Typically, either the finite element method (FEM) or finite-difference time-domain (FDTD) method is used. This type of simulator can use a so-called wave port, in which an area normal to the signal line — the port — is defined, and the electric field patterns of the relevant modes are calculated. Once the mode pattern is known, power is injected into the mode, and the reflected and transmitted powers to the various ports are calculated. S-parameters can be derived by looking at the power, as well as either the current or voltage on the signal line. Where is the ground in the case of voltage? The user is required to define an impedance line over which the electric field is integrated in order to get the voltage. The conductor on which the line starts is the ground for the port; i.e., it is the reference point for the voltage. The problem is that different ports can have completely different reference conductors, which can, to say the least, be confusing. For example, S-parameters can be calculated for a line going through several layers of a package, where the ports at the two ends of the line are referenced to completely different ground planes. More on this problem of multiple ground planes later.

The second group of simulators solves for the currents on the conductors using method of moments (MoM). The conductors are meshed, and the currents in each cell interact with every other cell through electric (capacitive) and magnetic (inductive) coupling, as defined by an object known as Green's function. A dense matrix equation results, which must be solved for the unknown currents on the meshes. In this type of simulator, an analogy to the wave port mentioned above is used: the edge port. The port is either placed at the end of the signal line or the edge of the simulation box. (MoM simulators differ in that some exist in an electrical box and some are in open space.) The port is excited, the incident and reflected currents are measured, and the S-parameters are determined. The calculation assumes the currents are operating in a transmission line type of mode, and a local ground return is assumed. Where is it? In the case of a boxed MoM simulator, it is the sidewall, and the voltage excitation is across a small gap between the signal line and the sidewall.

In the case of the unboxed simulator, either a ground plane (microstrip and striplines) or side grounds (coplanar lines) are assumed. It should be noted that the grounds can be different at the various edge ports, which can lead to problems with interpreting the S-parameters. Fortunately, in the case of a boxed simulator, the sidewalls represent a fairly good ideal ground. The bottom ground plane usually does the same for the microstrip and stripline cases (unless it is extremely lossy), but ultimately it is the job of the designer to determine if the grounds at the various ports can be considered the same. Usually, de-embedding algorithms are used for this type of port to remove the capacitive discontinuity of the voltage gap. This procedure is similar to calibration procedures for network analyzers.

Both types of simulators mentioned above support other types of ports. They have the advantage of being easy to use in the interior region of the circuit, unlike the wave or edge ports, which must be placed at the boundary of the circuit. These ports are analogous to using a probe in the laboratory, as opposed to the network analyzer's coaxial cable connection, and they are much easier to place around the circuit.

Figure 1 shows three popular types of ports: the edge port as previously discussed, the via port, and the series port. The via port can be viewed as a probe tip attached to the circuit through a small hole in the ground plane. The series port can be viewed as a two-point probe, with red and black probe tips. Where is the ground for these ports? In the case of the via port, it is the ground plane through which it protrudes. In the series port, it is the analogy of the black probe tip. In *Figure 1*, the series port is made by cutting the conductor and placing a resistor (the port impedance) across the gap created by the cut. The "+" side corresponds to where the red probe tip would attach; the "-" side corresponds to where the black probe tip would attach. This is the local ground for the port. Some simulators support placing the probe tips at arbitrary points in the circuit; electrically, they still behave exactly as the illustration.

Take care when using the series port. A good analogy would be that you are free to insert the two probe tips into the back of your television and get a difference of a few volts between them, but you SHOULD NOT reach your hand in. Fortunately in the case of EM simulation, the worst that will happen is that the S-parameter file will make no sense, as opposed to possible death in the case of the television. Normally, the series port is used for inserting a model of a small device, such as a transistor, into the simulation. It should be noted that these ports also have parasitics. The inductance of the via port is an example. These parasitics may or may not be de-embedded out, depending on the specific simulator.

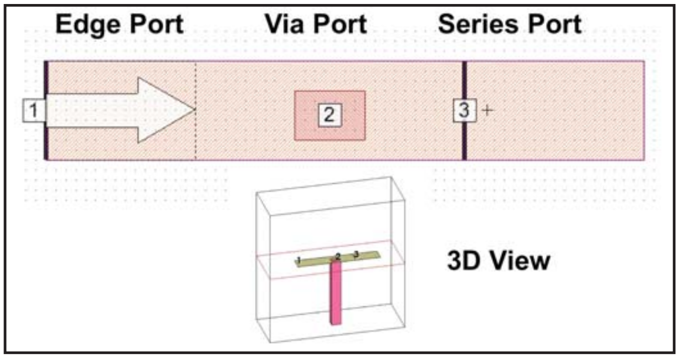

Figure 1: Edge, via port, and series ports

What is the grand conclusion? EM simulators use ports to get S-parameters. Those ports make an assumption of a local ground in some way. The local grounds at the various ports should electrically all be at about the same voltage, or interpretation of the S-parameter file by the circuit simulator could be problematic.

Modeling An Imperfect Ground

What is the course of action if the grounding structure used in the EM simulation is far from perfect? For example, the ground plane might be resistive. The local grounds used by the ports could have a large voltage difference between them, which in turn could lead to problems when using the S-parameter file in the circuit simulator. The answer is that ground must be modeled in some way.

Before delving into this topic, I should mention a technique that is occasionally attempted: exposing the ground of the S-parameter file. This is a perfectly valid concept when properly used, but unfortunately it often is misapplied.

For example, assume the user starts with a two-port S-parameter file. By exposing the ground, a three-port S-parameter file is obtained, with the third port being the local ground return. Remember, it has been assumed that the local grounds of the ports are electrically the same; i.e., they are connected by a very, very good ground return. The exposed node connects to that ground return. By doing this, for example, the engineer can DC (direct current) bias the ground return.

Typically, this procedure is used for transistor S-parameters. Transistors have three ports, but when measuring a transistor with a network analyzer, one port is grounded, and a two-port S-parameter file results. By exposing the ground node of the S-parameter file, the designer can attach elements to the previously grounded port, for example, an inductance to the common source node. This concept also can be used where the transistor is housed in a package and the global circuit ground is attached to the transistor package's ground. The key point is that this method works because the local grounds of the ports are essentially the same and are attached to each other by a perfect grounding structure.

A common mistake is to assume that imperfect ground properties can be observed by looking at the exposed node, for example, the loss of the ground plane. The exposed ground approximation assumes the ports' local grounds are the same. However, imperfect ground properties would make the ports' local grounds different voltages. There is even greater confusion with multiport S-parameter files, where differential ports, or local grounds, are requested. For example, a two-port S-parameter file will now have four ports, with port 3 corresponding to port 1's ground, and port 4 corresponding to port 2's ground. *Figure 2* illustrates the mathematical operation that is carried out. In electrical terms, a 1:1 transformer has been added to each port. In other words, the designer is forcing a differential mode at the ports.

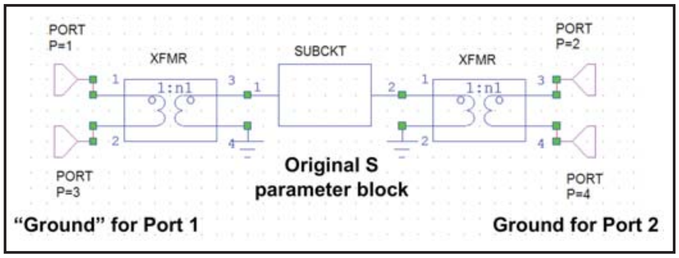

Figure 2: Exposed nodes — multiport

This can be a useful tool, but unfortunately, it is misunderstood by many designers. They incorrectly assume they are looking at the local ground of the port. For example, they think they can measure the resistance of the original lossy ground in the EM simulation by placing an ohmmeter across the two new ground ports. They can't. The original S-parameter file did not have this information (it assumed the local grounds were at the same voltage). These ports were created by the mathematical operation of adding transformers. A math trick cannot recover lost physics, no matter how hard the designer tries.

How does one model an imperfect ground? First, it already has been decided that the various ports' local grounds must all be connected by a very good ground in order to make sense of the S-parameter file. Therefore, the imperfect ground cannot be treated as a ground. It must be explicitly simulated as another conducting signal path with appropriate ports. Then, the designer can explicitly address the ground's properties. In other words, there is at least one very good ground for port references, and any number of imperfect power planes with their own ports. Following are some specific examples that illustrate different approaches to ground modeling.

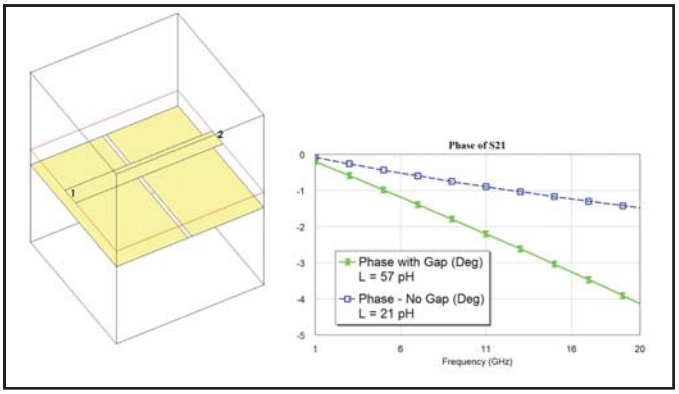

Figure 3: Gap in ground plane simulation

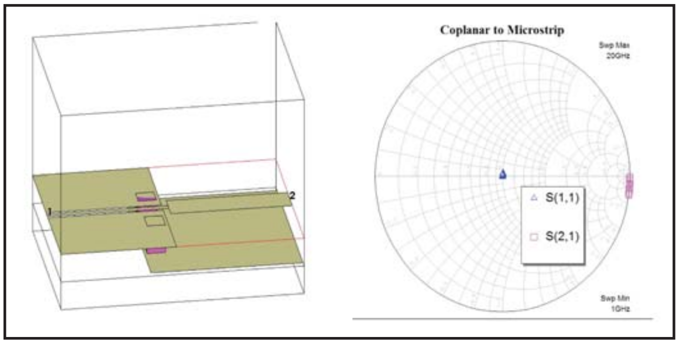

Figure 4: Coplanar-to-microstrip line transition

Examples Of Ground Modeling Issues

Example 1: A Gap In The Ground Plane

Figure 3 shows a simple gap in a ground plane. The line is on 100 um-thick alumina and is 100 um wide. It runs over a 20 um-wide gap in the ground plane. The S-parameters are shown. The gap adds inductance to the line and can easily be modeled by a lumped inductor (57 pH for this example). This example is straightforward, because the gap is localized with the ground currents able to go around it. A more difficult problem occurs when the current must go through decoupling capacitors or other discontinuities to get to the other side.

Example 2: A Coplanar-To-Microstrip Transition

The geometry is shown in *Figure 4*. The ground of the coplanar waveguide (CPW) has been connected to the ground of the microstrip by means of two vias and optimized to give a good transition to 20 GHz. Notice that the side walls of the simulator are the ground references for each port. Therefore, the two ports have the same ground reference, and the S-parameters can be sensibly interpreted.

Example 3: A Board-To-Package Transition

In this simulation, two signal lines and their grounds are attached to a chip by means of bondwires. There are five bondwires, in a G-S-G-S-G configuration, as shown in *Figure 5*. Therefore, the chip ground is not the same as the board ground. This effect is modeled by explicitly simulating the local chip ground as another signal plane. It is not attached to node 0. Rather, it is only attached to the grounding bondwires through the ports of the EM simulation. Notice that node 0 is the board ground in the simulation. The signal ports in the package are series ports (see the red and black probe tip analogy above). They are referenced to the local ground of the package in which the designer is interested. It is very important that the signal ports of the package do not go to node 0; they only know of the local package ground.

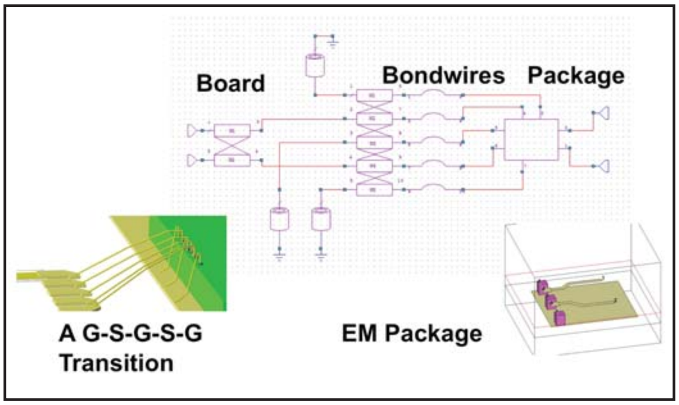

Figure 5: A board-to-package transition

To summarize, the ground transition is explicitly modeled as another signal net. Global ground is only used at one end, in this case, the board. The ports in the package are only referenced to local ground. The designer hopes, of course, that the local package ground is electrically close to the global ground.

A more sophisticated approach to working with a local ground is given in reference 3. The local ground is accounted for by performing an EM simulation from the global ground to the local ground and de-embedding the resulting imperfect ground structure. The resulting S-parameter file is thereby corrected for the local ground being different than the global ground.

Conclusion

Circuit simulators require a global ground; EM simulators don't worry about ground at all. It is therefore the designer's responsibility to ensure that EM results (S-parameters) are properly interpreted by the circuit simulator. This article has discussed ports in EM simulators and how ground can be implicitly used when the resulting S-parameters are transferred to the circuit simulator.

The issue of imperfect ground return in the EM simulation and how to model it has been addressed, and examples have been included to illustrate the concept of ground in EM simulation. It is important to remember that all ports in the EM simulator assume that something close by is the ground. If different grounds for different ports are used, care must be taken to properly interpret the results of the simulation. ■

References

1. Ramo, S.; Whinnery, J.R.; and Van Duzer, T. (1994) *Fields and Waves in Communication Electronics*, Wiley, 3rd Edition.
2. Marks, R.B.; and Williams, D.F. (1992) "A General Waveguide Circuit Theory," *NIST Journal of Research*, Vol. 97, No. 5, pp. 533-562.
3. Rautio, J.C. (2005) "De-embedding the effect of a local ground plane in electromagnetic analysis," *Microwave Theory and Techniques*, Vol. 53, No. 2, pp. 770-776.

Dr. John Dunn *is a senior engineering consultant at AWR, where he is in charge of customer training and university program development. He is an expert on EM modeling and simulation for high-speed circuit applications. Dr. Dunn has experience in industry and academia. Prior to joining AWR, he was head of the Interconnect Modeling Group for four years at Tektronix. Before entering the engineering industry, Dr. Dunn was a professor of electrical engineering for 15 years at the University of Colorado at Boulder, where he led a research group in the areas of EM simulation and modeling. Dr. Dunn received his BA in physics from Carleton College and his MS and PhD degrees in applied physics from Harvard University. He is a senior member of the IEEE.*