BOARD LEVEL SIMULATION SPECIALISTS

ICD Stackup Planner - offers engineers/PCB designers unprecedented simulation speed, ease of use and accuracy at an affordable price

- 2D (BEM) field solver precision
- Characteristic impedance, edge-coupled & broadside-coupled differential impedance
- Unique field solver computation of multiple differential technologies per stackup
- Heads-up impedance plots of signal and dielectric layers
- User defined dielectric materials library - over 16,250 materials up to 100GHz

ICD PDN Planner - analyze multiple power supplies to maintain low impedance over entire frequency range dramatically improving product performance

- Fast AC impedance analysis with plane resonance
- Definition of plane size/shape, dielectric constant & plane separation for each on-board power supply
- Extraction of plane data from the integrated Stackup Planner
- Definition of voltage regulator, bypass/decoupling capacitors, mounting loop inductance
- Frequency range up to 100GHz
- Extensive Capacitor Library – over 5,250 capacitors derived from SPICE models

iCD

In-Circuit Design Pty Ltd

LEARN MORE

www.icd.com.au

Electromagnetic Fields: Part 2

by Barry Olney
IN-CIRCUIT DESIGN PTY LTD | AUSTRALIA

SUMMARY: In last month's column, Barry Olney discussed how magnetic fields revolve around the earth and how these fields are also present in a multilayer board. Part 2 will look at how electromagnetic fields influence transmission lines and how they can be applied in a BEM field solver.

In last month's column, Electromagnetic Fields: Part 1, we looked at how magnetic fields revolve around the earth and how these fields are also present in a multilayer board. In Part 2, we will look at how electromagnetic fields influence transmission lines and how they can be applied in a BEM field solver.

Maxwell's Equations describe the relationship between electric and magnetic fields. These equations describe the field strength and current density within a closed-loop environment. They require extensive knowledge of higher-order calculus, so I will not bore you (or me) with further details. However, much insight into high-speed design can be gained by understanding the behavior of transmission lines and their associated electromagnetic fields.

In Figure 1, copper is poured around the differential pair. This configuration is commonly used in RF design and called coplanar wave guides. As mentioned in my previous column Ground Pours: To Pour or Not to Pour?, this is not a particularly good strategy for high-speed, digital design as it tends to alter (lower) the impedance of traces that run adjacent to a ground pour area. The traces of the differential pair are still coupled to each other and the ground plane, but are also coupled to the copper pour. This will reduce the impedance by ~25% and provide little additional shielding as the ground plane below is already used as a reference plane.

Figure 2 shows how the fields interact for two common stripline configurations: edge coupled differential (top) and offset, broadside coupled differential (bottom). In the top example, the traces are closely coupled to each other and also to the ground plane above. Since they are located much closer to that plane, the fields follow the path of least inductance rather than coupling to the plane further below (although it does have some influence).

In this case, the ground plane above will be used for the current return path of the signal. It is crucial to understand which plane will be used for the return current path. In general, for offset striplines, whichever plane is closest to the trace has the most influence on impedance.

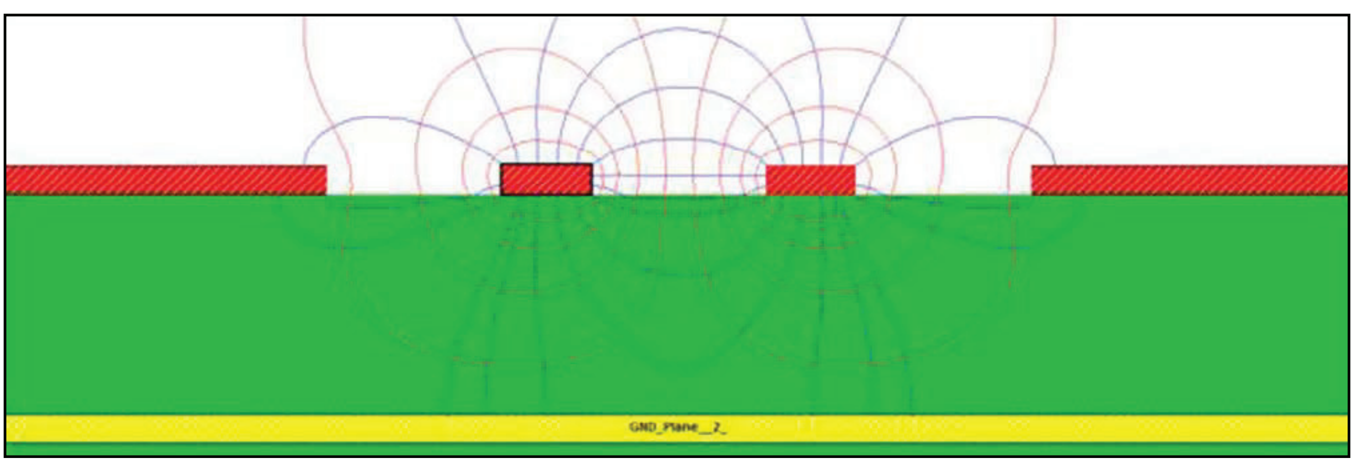


Figure 1: Differential pair as a coplanar wave guide.

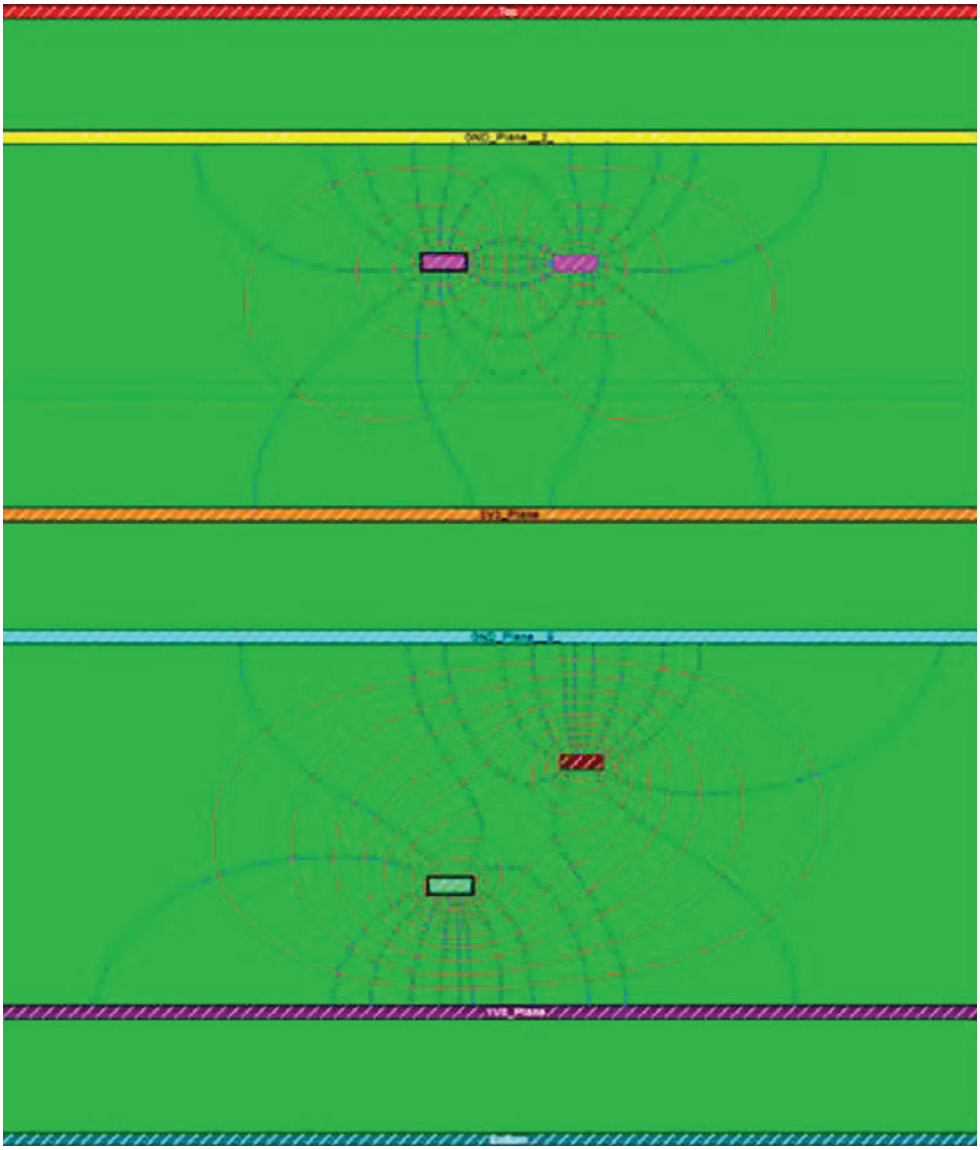


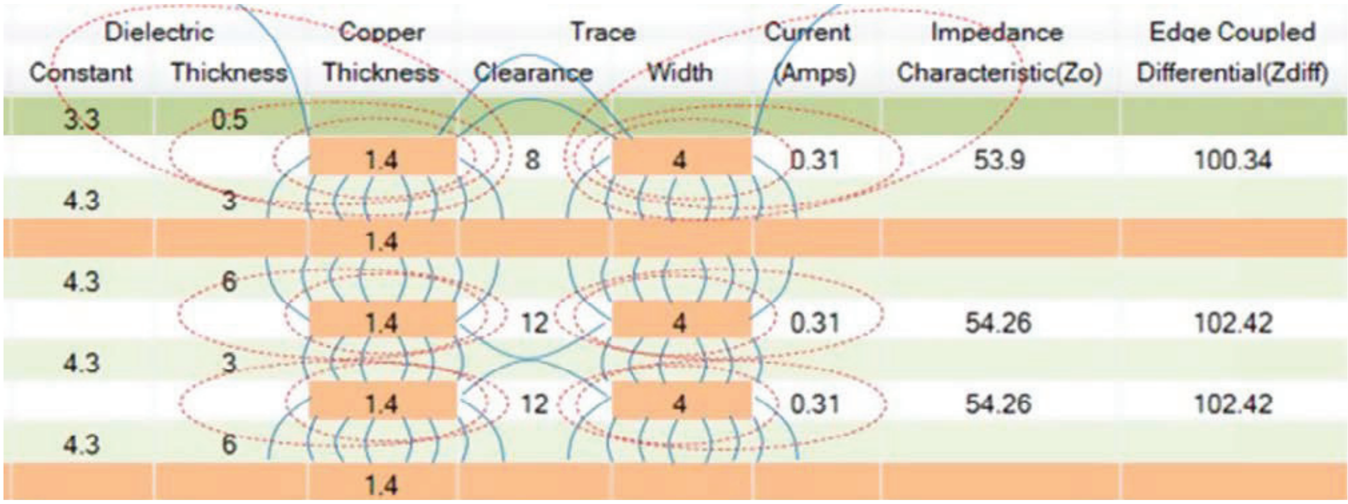
Figure 2: Edge-coupled differential (top) and broadside coupled differential (bottom).

Smack in the middle – both planes are equally important. But this would not be a good layout approach as both would also equally act as the return path. Therefore, a deliberate offset is advised to eliminate this possibility and to be absolutely sure that the correct plane will be used for the return path.

In the bottom example, each trace is coupled to the nearest plane and loosely (offset) broadside coupled to each other. This coupling should be avoided in multilayer boards as it causes crosstalk. It is best to route these two signal layers orthogonally or better still – only have one signal layer between planes to totally eliminate this issue.

The impedance of broadside coupled traces is affected by the mechanical registration of layers of the substrate during the fabrication process. IPC recommends layer-to-layer registration between any two adjacent layers to be within two mils. As you can imagine, the likelihood of two traces aligning perfectly is low. For instance, for a 4-mil trace, it could well be that there is only an overlap of two mils. One possible application where broadside coupling may have an advantage over edge coupled traces, is when routing an interleaved bus on a backplane – routing a number of differential pairs through a succession of connector pin fields where only a single trace fits between pins.

A boundary element method (BEM) field solver, such as that integrated in the ICD Stackup Planner, harnesses the field charges surrounding the traces to calculate an impedance matrix. The boundaries (both dielectric/metal and dielectric/dielectric) are split into many elements and each element is assumed to have a uniform charge density. Hence, small elements are needed where the charge density changes rapidly and larger elements where the charge density is more uniform. Defining the elements is as much an art as a science and this all impacts on the speed of simulation. Green's Theorem Method and the matrix inversion yield a solution of an integral equation.



Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Impedance Characteristic(Zo)	Edge Coupled Differential(Zdiff)
3.3	0.5						
		1.4	8	4	0.31	53.9	100.34
4.3	3						
		1.4					
4.3	6						
		1.4	12	4	0.31	54.26	102.42
4.3	3						
		1.4	12	4	0.31	54.26	102.42
4.3	6						
		1.4					

Figure 3: The electromagnetic fields as seen by the field solver in the ICD Stackup Planner.

To increase simulation speed, ICD has developed unique, proprietary algorithms to automatically adjust the solution space relative to the defined variables.

In Figure 3, I have roughly drawn in the fields that would be seen by the field solver software. A 16-element impedance matrix is required to extract all the values of the stripline configuration (layer 3 & 4) where there are two signal layers between the planes.

Equation 1

$$\begin{matrix} Z11 & Z12 & Z13 & Z14 \\ Z21 & Z22 & Z23 & Z24 \\ Z31 & Z32 & Z33 & Z34 \\ Z41 & Z42 & Z43 & Z44 \end{matrix}$$

$$\boxed{Zdbs} = Z11 - 2{*}Z13 + Z33$$
$$\boxed{Zdiff} = Z11 - 2{*}Z12 + Z22$$
$$\boxed{Zdiff} = Z33 - 2{*}Z34 + Z44$$
$$\boxed{Zdbs} = Z22 - 2{*}Z24 + Z44$$

where Zdiff is edge coupled and Zdbs is broadside coupled differential impedance

The impedance matrix gives the impedance of the system of coupled nets in the coupling region. The values in the diagonal matrix positions (example Z12 and Z21) can be thought of as giving the impedances to ground of the corresponding nets, accounting for the presence of the other nearby, coupled traces. When an IC drives into one of the lines, however, it "sees" not only the diagonal impedance for that line, but also some of the off-diagonal terms in the matrix.

For traces that are only weakly coupled, the diagonal impedance terms are dominant, and the diagonal values are close to what they would be if the lines were completely isolated from each other. As the coupling becomes stronger, the diagonal terms deviate more from their standalone values, and the off-diagonal terms increase.

If the configuration is microstrip or embedded microstrip (rather than stripline), and the electromagnetic fields they generate lie in a mixture of dielectrics (e.g., FR-4 and air), then multiple propagation velocities exist per line.

Much insight into high-speed design can be gained by understanding the behaviour of transmission lines and the influence of their associated electromagnetic fields. Controlled impedance design can be simplified and the path of the return current can be visualized by understanding the field coupling.

Points to remember:

• Maxwell's Equations describe the relationship between electric and magnetic fields. Green's Theorem method and the matrix inversion yield a solution of an integral equation.

• Return current paths follow the path of least inductance rather than coupling to a plane further away.

• For offset striplines, whichever plane is closest to the trace has the most influence on impedance. A deliberate offset is advised to eliminate this possibility and to be absolutely sure which plane will be used for the return path.

- The impedance of broadside coupled traces is affected by the mechanical registration of layers, of the substrate, during the fabrication process and should be avoided.
- A BEM field solver, such as that integrated in the ICD Stackup Planner, harnesses the field charges surrounding the traces to calculate an impedance matrix.
- The impedance matrix gives the impedance of the system of coupled nets in the coupling region. **PCBDESIGN**

References

1. Advanced Design for SMT Course – Barry Olney

2. Beyond Design: Embedded Signal Routing – Barry Olney

3. Beyond Design: Ground Pours: To Pour or Not to Pour? – Barry Olney

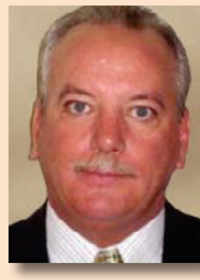
4. Beyond Design: Controlling the Beast – Barry Olney

5. Digital Transmission Lines – Ken Granzow

6. EMC and the PCB – Mark Montrose

7. The ICD Stackup and PDN Planner can be downloaded from www.icd.com.au

8. All trademarks are registered by their respective owners.



Barry Olney is managing director of In-Circuit Design Pty Ltd (ICD), Australia. The company is a PCB design service bureau that specializes in board-level simulation, and is the developer of the ICD Stackup Planner and ICD PDN Planner software. Contact Barry here.