Beyond Design: Not All PCB Substrates Are Created Equal

by Barry Olney | In-Circuit Design Pty Ltd | Australia

PCB substrates are all around us in every gadget we use. Be it a computer, a smart phone or a simple child's toy. The substrate may be rigid or flexible or a combination of both. It is a carrier for the electronic devices and the signal and power interconnects and is usually planar in structure with conductors separated by insulating dielectric materials. However, each product has a specific performance requirement and as such may need a distinct type of substrate to comply with the product's specifications particularly for high-speed designs.

Because the impedance of transmission lines is a function of substrate dielectric constant, essential requirements for a multilayer PCB dielectric material is extremely tight tolerance and consistency in the dielectric constant and in the thermal coefficient of dielectric constant (the amount of change in the dielectric constant as a function of temperature). In addition, the coefficient of thermal expansion (CTE), especially in the z-axis (through the thickness of the material), is of particular importance in multilayer designs because plated through holes (PTHs) are used to make connections between different layers of the stackup. The CTE is a yardstick for expected PTH barrel reliability.

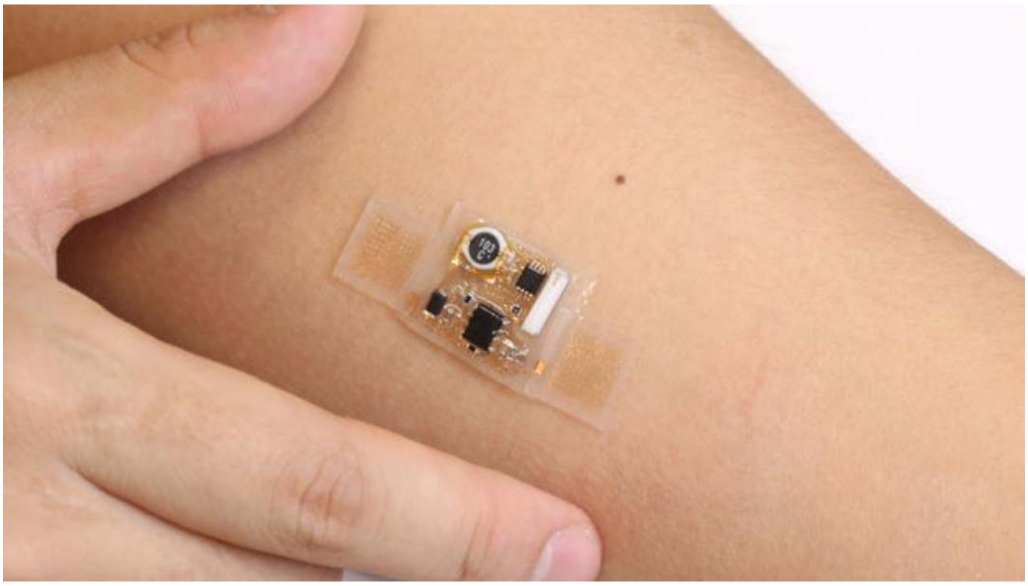


Figure 1–Biosensor patch worn on the skin (source CBS News)

A wire suspended in free air has impedance in the order of 330 ohms. This impedance may have been viable in the old vacuum tube days, but is not a good starting point for any digital design. A twisted pair cable, with an outer shield, has 100 ohms differential impedance. A coax cable with an inner wire surrounded by dielectric material and an outer shield is typically 50 to 75 ohms. So as we add a coupled return path, close to the signal conductor in the presence of a dielectric, the impedance reduces.

Typically for a digital design, a characteristic impedance of 40-60 ohms and differential impedance of 80-120 ohms are used. And, this becomes more important as the edge rates become faster. Also different technologies have their specific requirements. For instance, USB requires 90 ohms differential impedance and DDR3/4 require 40/80 ohms single ended/differential impedance. For perfect transfer of energy, the impedance of the driver must match the impedance of the transmission line. A good transmission line is one that has constant impedance along the entire length of the line, so that there are no mismatches resulting in reflections.

Technology is advancing rapidly in the field of medical electronics, providing doctors with more efficient ways to gather information. Figure 1 shows an ultra thin and wireless adhesive biosensor that is stuck to the skin. This device has the ability to monitor a person's heart rate and other vitals and transmit the data in real-time to a smart phone or computer. In this case, there appears to be no reference plane but the impedance, of the critical RF traces, can still be managed by the careful planning of coplanar structures on the surface of the flexible substrate. The coplanar impedance is determined by the ratio of trace width to clearance, so size reduction is possible without limit, the only penalty being higher losses. In addition, a virtual ground plane exists between any two adjacent traces, as there is no field at that point. Hence crosstalk effects, between adjacent traces, are very weak.

There is a common misconception that digital signals are transferred in the copper conductors of a multilayer PCB substrate. This is ‘flat-earth’ thinking. A transmission line does not carry the signal itself but rather, guide’s electromagnetic energy from one point to another through the substrate. Voltage and current do exist in the conductor, but only as a consequence of the field being present as it moves past. The path should also control the characteristic impedance so there are minimal reflections. What we really need to do is to provide a smooth, consistent path for the flow of electromagnetic energy.

The speed of a computer does not depend intrinsically on the speed of electrons, but rather on the speed of energy transfer between electronic components. The dielectric material determines the velocity (v) of propagation of the electromagnetic (EM) energy:

$$v = \frac{c}{\sqrt{Er}}$$

where the speed of light (c) is 3x10^8 m/s.

With a typical Er (aka Dk) of 4 for FR-4 material, the signal will travel at approximately half the speed of light (*c/2*) through the substrate regardless of the clock frequency. The lower the Er–the faster the speed. The signals, with their relative timing requirements, essentially ride the EM carrier wave. So, matching the propagation speed, between signals on different stackup layers, is crucial to ensure the correct timing margin at the receiver.

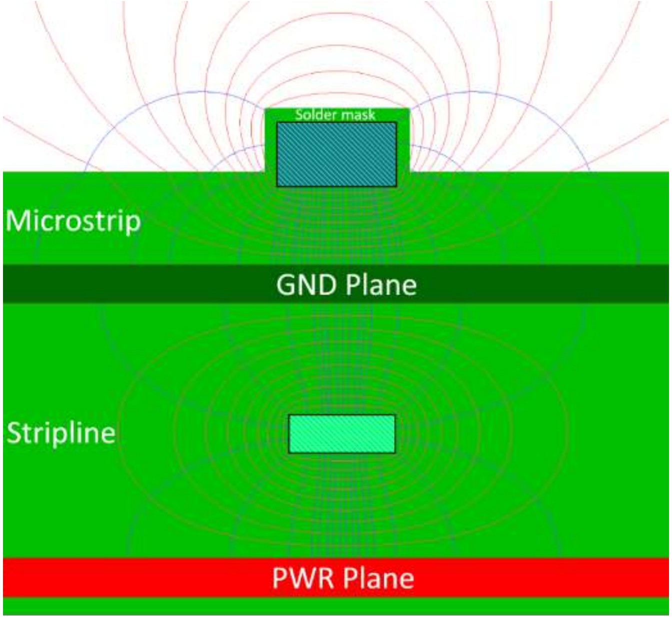


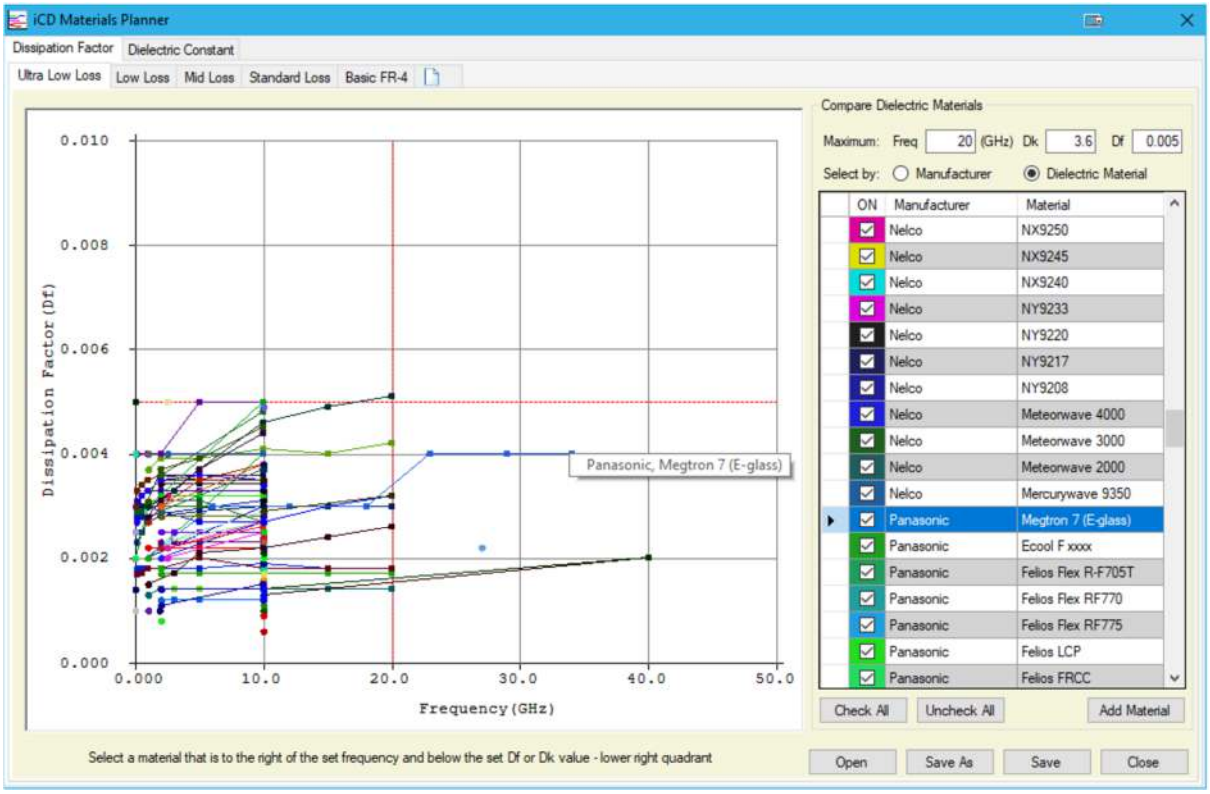
Figure 2–Microstrip and stripline electromagnetic fields (simulated in HyperLynx)

A stripline is any trace sandwiched between reference planes on both sides as in Figure 2. The electric fields (blue) of a stripline are totally contained between the two solid planes, so the speed of propagation for signals, guided by the trace, is entirely determined by the dielectric constant of the surrounding materials.

On the other hand, a microstrip is any trace fabricated on the outer layers of a PCB. A microstrip has dielectric material and a plane on one side and air on the other. An embedded microstrip is similar but is covered in a conformal coating such as solder mask or another dielectric material. In this case, the effective dielectric constant should be calculated by a field solver and represents a combination of the surrounding materials. There are also other variants of microstrip and stripline; such as build-up microstrip, coplanar waveguides and dual (a)symmetric stripline.

The electromagnetic fields surrounding the microstrip exist partially within the dielectric material(s) and partially within the surrounding air. Since air has a dielectric constant of one, which is always lower than that of FR-4 and solder mask, mixing a little air into the equation will speed up the signal propagation. Even if the trace widths are adjusted on each layer, so that the impedance is identical, the propagation speed of microstrip is always faster than stripline–typically by 13-17%.

At frequencies above 1GHz the Dissipation Factor (Df), another selection criteria for high-speed PCB substrate material, comes into play. Df is a parameter of a dielectric material that quantifies its inherent dissipation (loss) of electromagnetic energy. It refers to the tangent of the angle in a complex plane between the resistive (lossy) component of an electromagnetic field and its reactive (lossless) component. Standard FR-4 has a Df of ~0.02 whereas an ultra low loss dielectric may have < 0.005 at 10GHz.



Maximum: Freq 20 (GHz) Dk 3.6 Df 0.005

Select by: ○ Manufacturer ● Dielectric Material

ON	Manufacturer	Material
☑	Nelco	NX9250
☑	Nelco	NX9245
☑	Nelco	NX9240
☑	Nelco	NY9233
☑	Nelco	NY9220
☑	Nelco	NY9217
☑	Nelco	NY9208
☑	Nelco	Meteorwave 4000
☑	Nelco	Meteorwave 3000
☑	Nelco	Meteorwave 2000
☑	Nelco	Mercurywave 9350
☑	Panasonic	Megtron 7 (E-glass)
☑	Panasonic	Ecool F xxxx
☑	Panasonic	Felios Flex R-F705T
☑	Panasonic	Felios Flex RF770
☑	Panasonic	Felios Flex RF775
☑	Panasonic	Felios LCP
☑	Panasonic	Felios FRCC

Select a material that is to the right of the set frequency and below the set Df or Dk value - lower right quadrant

Figure 3–Loss profile for ultra low loss dielectric materials (source iCD Materials Planner)

Dielectric constant and dissipation factor contribute to the frequency dependent loss and to degrade the bandwidth and speed of the signal. The signal quality transmitted through the medium, and picked up at the receiver, will be affected by any impedance discontinuities and by the losses of the dielectric materials. Fortunately, high frequency dielectric materials generally have low Er and low Df enabling the signals to propagate faster; have less loss and therefore higher bandwidth. Figure 3 shows the loss profile of selected dielectric materials in the iCD Materials Planner.

When we design a transmission line, as part of a multilayer stackup, we are not just defining the copper traces–we are also specifying the dielectric to transfer the electromagnetic energy. The traces and vias guide the energy through the substrate. A field solver uses the combined effects of trace width, clearance and thickness plus the dielectric constant and material thickness to determine the impedance of the trace. However, the speed of propagation is independent of trace geometry and is totally determined by the dielectric material. All dielectric materials that compose a substrate have different properties, so one needs to carefully select the optimal materials for the required purpose.

Key Points:

- The impedance of transmission lines is a function of substrate dielectric constant.
- The CTE is a yardstick for expected PTH barrel reliability.
- Adding a coupled return path, close to the signal conductor in the presence of a dielectric, reduces the impedance.
- Typically for a digital design, a characteristic impedance of 40-60 ohms and differential impedance of 80-120 ohms are used.
- A good transmission line is one that has constant impedance along the entire length of the line, so that there are no mismatches resulting in reflections.
- Coplanar impedance is determined by the ratio of trace width to clearance, so size reduction is possible without limit, the only penalty being higher losses.
- A transmission line does not carry the signal itself but rather, guide's electromagnetic energy from one point to another through the substrate.
- The signals, with their relative timing requirements, essentially ride the EM carrier wave.
- The speed of a computer does not depend intrinsically on the speed of electrons, but rather on the speed of energy transfer between electronic components.
- The propagation speed of microstrip is always faster than stripline–typically by 13-17%.
- Dissipation Factor is a parameter of a dielectric material that quantifies its inherent dissipation (loss) of electromagnetic energy.
- Dielectric constant and dissipation factor contribute to the frequency dependent loss and to degrade the bandwidth and speed of the signal.
- High frequency dielectric materials generally have low Er and low Df enabling the signals to propagate faster; have less loss and therefore higher bandwidth.

References:

Beyond Design: Microstrip Coplanar Waveguides, Controlled Impedance Design, Faster than a Speeding Bullet, Signal Flight Time Variance in Multilayer PCBs, Material Selection for SERDES Design, It's a Material World–Barry Olney
Microwaves and RF: Picking Materials for Multilayer PCBs
Tiny biosensor shows big promise–CBS News
Fast Circuit Boards: Energy Management–Ralph Morrison

Bio:

Barry Olney is Managing Director of In-Circuit Design Pty Ltd (iCD), Australia. The company developed the iCD Design Integrity software incorporating the iCD Stackup, PDN and CPW Planner, is a PCB Design Service Bureau and specializes in board level simulation. The software can be downloaded from www.icd.com.au