Beyond Design – Material Selection for Digital Design

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

In a previous column, Material Selection for SERDES Design, I pointed out that materials used for the fabrication of the multilayer PCB absorb high frequencies and reduce edge rates and that loss, in the transmission lines, is a major cause of signal integrity issues. But we are not all designing cutting edge boards and sometimes we tend to over specify requirements that can lead to inflated production costs. In this column, I will look at what types of materials are commonly used, for digital design, and how to select an adequate material to minimize costs. Of course selecting the best possible material will not hurt – but may blow out the costs.

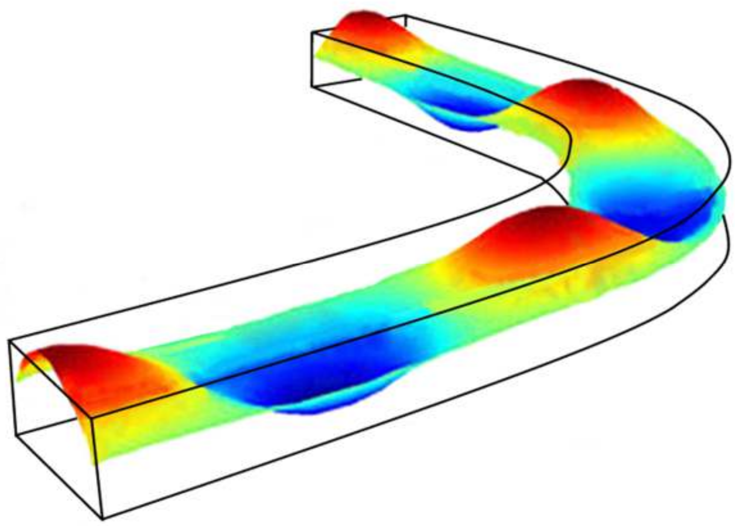

Figure 1 – Simulated signal propagating through a curved waveguide[5]

Signals propagate in a vacuum or air at – or about – the speed of light. But, as the electromagnetic energy is enveloped in a dielectric material, sandwiched between planes in the PCB medium, it slows down. Figure 1, illustrates a signal propagating through a curved waveguide. This is representative of a typical stripline configuration of a PCB. What needs to be understood is that the signal traces, on a PCB, simply guide the signal wave – as the electromagnetic energy propagates in the surrounding dielectric material. It is the dielectric material that determines the velocity (v) of propagation of the electromagnetic energy:

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

Equation 1

Keep in mind that c is the speed of light (in free space) and Er is the dielectric constant of the material (FR-4 is ~4.0). By contrast, the Er of air is 1. Therefore, the velocity of propagation in FR-4 is about half the speed of light or 6 inches per ns. The important concept is that it is the electromagnetic energy that propagates down the transmission line – not electron flow. Electrons flow at about 0.4 inches per second – snail pace by comparison.

The electrical properties of a dielectric material can be described by two terms:

1. The Dielectric Constant or Relative Permittivity (Er or Dk) is the ratio of the amount of electrical energy stored in a material by an applied voltage. It describes how the material increases the capacitance and decreases the speed in the material.
2. The Dielectric Loss or dissipation factor/loss tangent (Df) is a parameter of a dielectric material that quantifies its inherent dissipation of electromagnetic energy.

Dielectric constant and dielectric loss are not a function of the geometry of the transmission line – they are a function of the dielectric material in which the signal propagates, their distribution in the PCB stackup and the applied frequency. These mechanisms contribute to the frequency dependent loss and to degrade the 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. The glass epoxy material (FR-4), commonly used for PCB's, has negligible loss for digital applications below 1 GHz. But, at higher frequencies the loss is of greater concern.

So if you have a fast rise/fall time, high frequency signal, then the wave needs to propagate at higher speed and therefore the Er needs to be low to enable this. If a material with a high dielectric constant is placed in an electric field, the magnitude of that field will be measurably reduced within the volume of the dielectric. Therefore, a lower Er is desirable for high-speed design.

It is best to use the value of the dielectric constant applicable at the highest frequency of interest. For digital signals, the highest frequency of interest (f) depends on the rise/fall time (Tr) and is approximated by:

$$f = \frac{0.5}{Tr}$$

Equation 2

Therefore, for a 1ns rise time signal, the frequency of interest will be 500MHz. But then, the maximum bandwidth also needs to consider the 3rd or 5th harmonic of the fundamental – 1.5GHz to 2.5GHz – in this case. The bandwidth is an indication of the highest data rate that can be transmitted by an interconnect. So for a 1ns rise time signal, we should look at about a 2GHz material.

Also of importance is the Glass Transition Temperature (Tg), which is the point at which a glassy solid changes to an amorphous resin/epoxy. If the reflow temperature exceeds the Tg for an extended period, the material rapidly expands in the z-axis. Plus mechanical material properties degrade rapidly – strength and bonds in the material. A high Tg guards against barrel cracking and pad fracture during reflow. Standard FR-4 has a Tg of 135-170C, whereas the high-speed materials are generally over 200C.

There a basically two types of dielectric material:

1. Woven fiberglass reinforced dielectric.
2. Fiberglass free dielectric.

At high frequencies, a non-uniform dielectric in the substrate can cause skew in differential signals. The inconsistency of the dielectric material comes from the fact that the fiberglass and the epoxy resin, that make up typical PCB core (laminate) and prepreg materials, have a different dielectric constant. And, because the fabricator cannot guarantee the placement of the fiberglass with respect to the location of the traces, this results is uncontrolled differential skew. A fiberglass free material can be used to eliminate differential skew. However, fiberglass free materials come at a price. So for a cost effective solution, let's eliminate the fiberglass free dielectric.

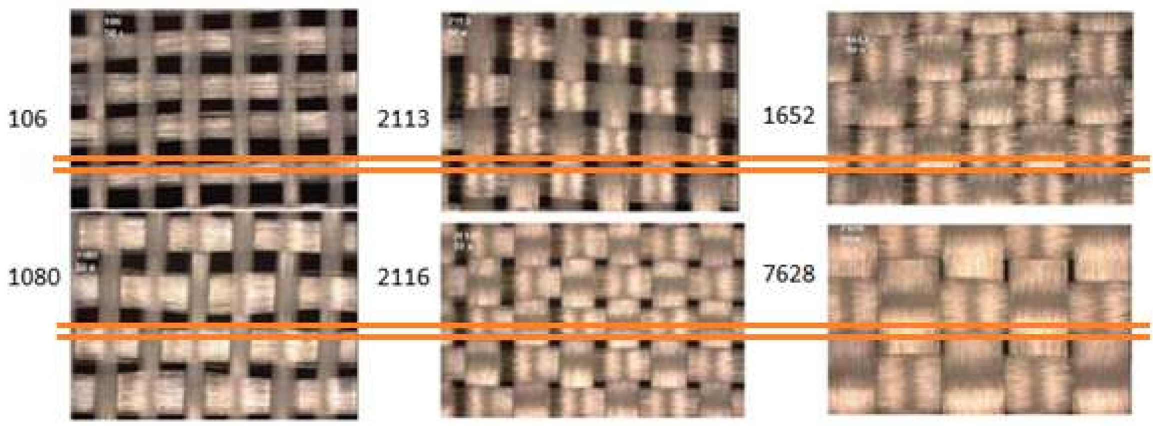

Figure 2 – Different types of fiberglass weaves compared to a differential pair – source Altera

Close attention should also be paid to the skew associated with the fiber weave effect. For high-speed data rates of 5 Gbps and above, this skew significantly cuts into the available jitter unit interval (UI) budget and leads to a reduction in the observed eye width at the receiver. If the flexibility exists, specify a denser weave material (2113, 2116, 1652 or 7628) compared to a sparse weave (106 and 1080). Figure 2, compares the different types of fiberglass weaves to a 4/4 mil differential pair. Notice that one side of the pair can be routed over the fiberglass and the other over the gap (resin), depending on the placement. The different dielectric constants create skew. However, routing the differential signals diagonally across the weave can reduce this skew considerably.

Typically, when the impedance of a substrate it first calculated, "virtual materials" are used as the basis. In other words, we choose a round number to represent the dielectric constant, dielectric thickness, and the attributes of the trace thickness and width to establish a solution. However, these are not the attributes, of the actual materials, used by the Fab shop to manufacture the board and are inherently inaccurate. I'm not saying that the use of virtual materials should be avoided but rather, the numbers need to be in the ball park to begin with.

In order to select the correct dielectric materials and variables for your substrate, you need to consider the following:

1. Dielectric Loss needs to be low.
2. Dielectric Constant needs to be low.
3. Glass Transition Temperature needs to be high (=> 180^{o} C).
4. Dielectric Thickness needs to be low.
5. Trace thickness, width and separation need to be above the manufacturable limits. Trace width/clearance should not go below 4/4 mils to minimize costs.
6. And most important of all, the price needs to be low.

All of the above need to be considered, to establish the right material without over-design.

UNITS: mil — 7/3/2014 — Total Board Thickness: 62 mil

Differential Pairs > 50/100 ohms | USB 90 ohms

Layer No.	Via	Description	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Characteristic Impedance (Zo)	Edge Coupled Differential (Zdiff)	Broadside Coupled Differential (Zdbs)
1	8	Plane	GND	Conductive			1.4						
		Core		Dielectric	3.9	4							
2		Signal	Inner 3	Conductive			1.4	10	4	0.31	54.05	99.19	86.89
		Prepreg		Dielectric	3.9	10							
3		Signal	Inner 4	Conductive			1.4	10	4	0.31	54.05	99.19	86.89
		Core		Dielectric	3.9	4							
4		Plane	VDD	Conductive			1.4						

Figure 3 – Stripline configuration using "virtual materials"

Once the ball park, virtual material numbers are established, the material needs to be selected for 2GHz operation. This I suggest you do in consultation with your preferred Fab Shop, as choosing the materials that they stock will result in up to 5% better accuracy. Obviously what you select is based on what is available at a reasonable price. The ICD Stackup Planner has a massive 16,900 materials up to 100GHz to choose from, so all this information is right at your fingertips. Boolean searches can be done in order to reduce the select list as Illustrated in Figure 4. Look for a 2GHz material with Er<4, Df<0.02 and Tg=>180C. In Figure 5, I have chosen ITEQ IT-180A which fits the specs.

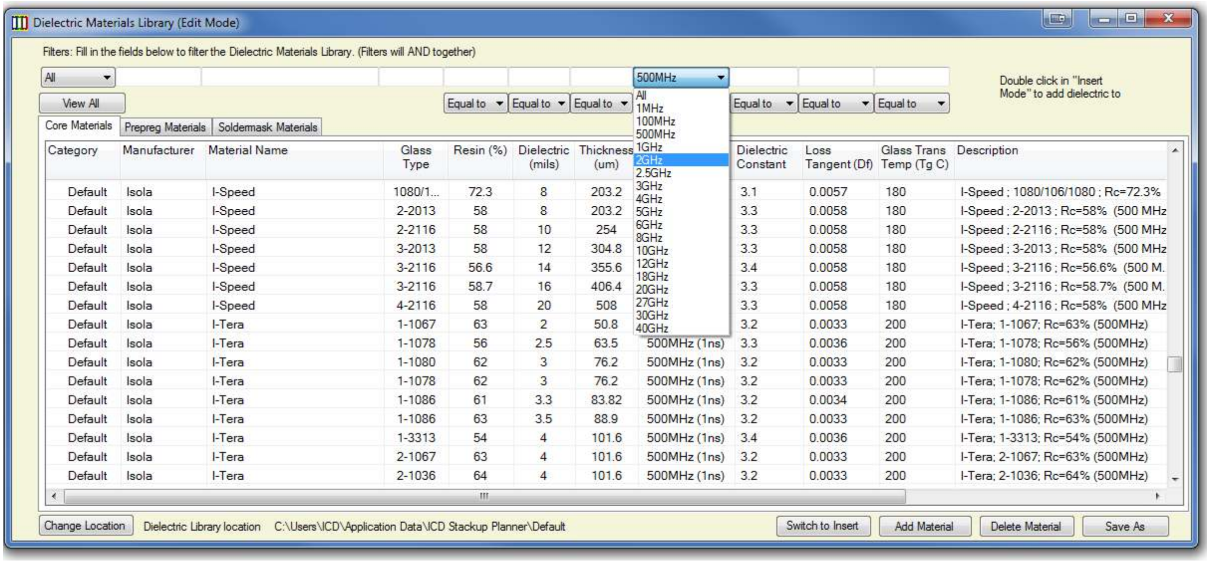

Figure 4 – Material selection in the ICD Stackup Planner 8,800 part Dielectric Materials Library

Prepreg materials are only available up to 8 – 9 mil thick, so in order to attain the desired thickness, multiple prepregs must be stacked together to give the required 10 mils. In this case, I have use 2 x 2.8 plus a 4.6 giving 10.2 mils total. Make sure these are symmetrical, about the center, otherwise there will be a slight offset in impedance due to the field solver seeing an imbalance in dielectric constant.

UNITS: mil — 7/3/2014 — Total Board Thickness: 62 mil — Differential Pairs > 50/100 ohms | USB 90 ohms

Layer No.	Via	Description	Layer Name	Material Type	Dielectric Constant	Dielectric Thickness	Copper Thickness	Trace Clearance	Trace Width	Current (Amps)	Characteristic Impedance (Zo)	Edge Coupled Differential (Zdiff)	Broadside Coupled Differential (Zdbs)
1	8	Plane	GND	Conductive			1.4						
		Core		IT-180A; 1-2116; (2GHz)	3.8	4.0							
2		Signal	Inner 3	Conductive			1.4	10	4	0.31	54.45	99.73	87.82
		Prepreg		IT-180A; 106; Rc=62%; (2GHz)	3.9	2.8							
		Prepreg		IT-180A; 106; Rc=53%; (2GHz)	4	4.6							
		Prepreg		IT-180A; 106; Rc=62%; (2GHz)	3.9	2.8							
3		Signal	Inner 4	Conductive			1.4	10	4	0.31	54.45	99.73	87.82
		Core		IT-180A; 1-2116; (2GHz)	3.8	4.0							
4		Plane	VDD	Conductive			1.4						

Figure 5 – Virtual materials substituted with ITEQ IT-180A material

In conclusion, selecting an adequate material for the project will minimize the cost. The designer should calculate the highest frequency of interest, taking the bandwidth into account, then choose a dielectric material with the lowest Er (Dk) and Df with a Tg about 180C. And remember, choosing the materials that are stocked by your Fab Shop will result in up to 5% better accuracy.

Points to Remember

- Selecting the best possible material will not hurt – but may blow out the costs.
- Signals propagating in the dielectric material of a PCB slow down.
- Signal traces, on a PCB, simply guide the signal wave – as the electromagnetic energy propagates in the surrounding dielectric material.
- The velocity of propagation in FR-4 is about half the speed of light or 6 inches per ns.
- Dielectric constant and dielectric loss are not a function of the geometry of the transmission line – they are a function of the dielectric material in which the signal propagates, their distribution in the PCB stackup and the applied frequency.
- A low Er is desirable for high frequency design.
- It is best to use the value of dielectric constant applicable at the highest frequency of interest. However, the maximum bandwidth also needs to consider the 3rd or 5th harmonic of the fundamental.
- A high Tg guards against barrel cracking and pad fracture during reflow.
- A fiberglass free material can be used to eliminate differential skew but is costly.
- In order to select the correct dielectric materials and variables for your substrate, you need to consider dielectric constant and loss, glass transition temperature, trace thickness, width and separation and of course cost.
- Choosing the materials that are stocked by your Fab Shop will result in up to 5% better accuracy.
- The ICD Stackup Planner has a massive 8,800 materials up to 40GHz to choose from so all this information is right at your fingertips.

References:

1. Material Selection for SERDES Design – Barry Olney
2. Beyond Design: Transmission Line - From Barbed Wire to High-speed Interconnect – Barry Olney
3. Beyond Design: Matched Length Does Not Always Equal Matched Delay – Barry Olney
4. Beyond Design: Mythbusting - There are no One-way Trips! – Barry Olney
5. http://www.frankswebspace.org.uk/ScienceAndMaths/physics/physicsGCE/radioComms.htm
6. Electromagnetic Compatibility Engineering – Henry Ott
7. High-speed Signal Propagation – Howard Johnson
8. Simbeor Application Notes – Y. Shlepnev, Simberian Electromagnetic Solutions
9. The ICD Stackup Planner and PDN Planner can be downloaded from www.icd.com.au

Bio - Barry Olney is CEO of In-Circuit Design Pty Ltd (ICD), Australia. The company developed the ICD Stackup Planner and ICD PDN Planner software, is a PCB Design Service Bureau and specializes in high-speed, board level simulation.