Practical High Speed Design Part 3 of 3 Q & A

Questions / Comments	Answers / Response
When laying out the connectors with the power and ground at the ends of the connector head, can this be a typical approach if you're not doing high speed design?	Keep in mind that even with slow clocks, the edge rates can still consist of high frequencies. This is the result of the shrinking of the IC chips. For example, the 7400 series of today is on a much smaller die than the one 30 years ago. Though they will try to keep the characteristics the exact same, the rising edge will be much sharper. This sharp rising edge is a sum of high frequencies. As a result, a 10MHz design can still be impacted by HS frequency issues.
What preferred high-speed analysis tools do you use?	We use iCD (In-Circuit Design) which we represent here in North America and HFSS from Ansys.
In talking about glass weaves, is routing diagonally to the weave stilled practiced?	Yes. By routing diagonally, it allows for the use of a cheaper board material for those who are budget constrained.
Can copper pours on signal layers cause changes in impedance on adjacent traces and therefore problems?	Yes, the polygon will impact the trace impedance if there is insufficient spacing between the traces and polygon pours. Follow the 3W rule (spacing is 3 times the dielectric height).
In a 4 layer SGPS* design, a signal on top needs to use a via to continue to the bottom. How is the loop area affected by the transition from return reference to power reference? *SGPS: Layer 1 - Signal Layer 2 – Ground Layer 3 – Power Layer 4 - Signal	The loop area concern can be negated by using stitching capacitors at the transition of reference planes.
One layer of glass or two between layers? Sometimes 1080 is difficult to avoid.	Typically, the layer counts of glass are more controlled by material availability to create desired stackup than anything else unless the designer makes the specification otherwise.
What interconnect cable do you recommend for interfacing between 2 separated PCB boards over 8 inch distance? The signal bus is medium speed	I don't have a specific cable in mind. However, any cable used such that the impedance (Zo) can be maintained is a good cable. Cables are made from wire that is stretched/drawn so roughness is not as much a problem.