

BBG Tech Tips

Controlled Impedance

More and more of today's PCB designs demand faster processing of information requiring the bare board to have what is known as **controlled impedance**—the elimination or minimizing of discontinuities in the signal that a trace will deliver.

It is no longer a trace or track that connects a plated through hole to a via or to another device, but rather **transmission lines** designed to transport energy at high speeds with little loss in signal shape, magnitude or speed. This high speed circuit is similar to a high speed highway where controlled impedance prevents the formation of pot holes or speed bumps in the road.

Controlled impedance adds another level to the design, artwork, material selection and the PCB manufacturing process. Even solder mask, with its insulation properties will affect impedance values. Upon receipt of the order, the vendor will conduct a simulation to verify that the design will allow for the final impedance values with a usual tolerance of +/- 10%. If the design calculations do not agree, the PCB manufacturer will notify the customer to allow a change in the stack-up to meet the designed needs. Artwork being plotted has to be compensated to allow for the plating tolerances of the PCB manufacturer. Material consistency is also important—glass style, resin content and resin flow affects the dielectric constant which will affect the impedance results—usually requiring the same material to be used on repeat orders.

All controlled impedance boards will require TDR (Time Domain Reflectometry) measurements that confirm the impedance values are within tolerance. TDR traces are placed on a coupon that is located on the manufacturing panel. The coupon usually contains several traces of some length (usually up to 8 inches long) that run parallel to each other with a plated through hole at either end. Using a TDR measuring device, the coupon is verified that the impedance values are met. The testing is done on the coupon and is considered the benchmark that both designer and fabricator agree upon. The PCB manufacturer has no control over the discontinuities in the design meaning that if the coupon is correct so must be the PCB. Upon request, the TDR report along with the coupons may accompany the shipment.

Research for the above information may be from, but is not limited to, IPC reference manuals, the PCB Handbook, the Bare Board PWB Design Manual and consultation with industry professionals. Please consult a process engineer familiar with your company's PCB assembly process before making any procedure changes.