

Questions & Answers
What Does a 50 Ohm Impedance Look Like?

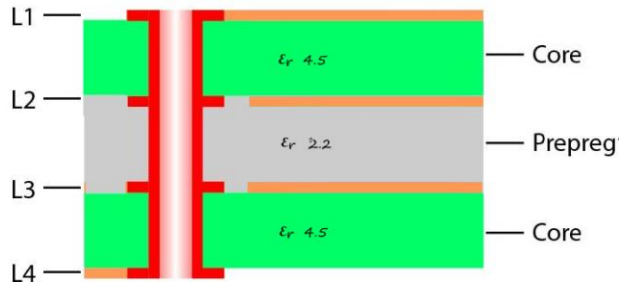
Question / Comment	Reply
<p>Was not clear to me from where comes the 50 Ohms impedance as well, I mean, why is not 40 Ohms for example?</p>	<p>Excellent question. It really is arbitrary. The first designers (mostly of cables) picked a standard because of material, loss, power transfer and manufacturing capability.</p> <p>The follow comes from Harmon Banning of W.L. Gore & Associates, Inc. cable:</p> <p><i>There are probably lots of stories about how 50 ohms came to be. The one I am most familiar goes like this. In the early days of microwaves - around World War II, impedances were chosen depending on the application. For maximum power handling, somewhere between 30 and 44 Ω was used. On the other hand, lowest attenuation for an air filled line was around 93 Ω. In those days, there were no flexible cables, at least for higher frequencies, only rigid tubes with air dielectric. Semi-rigid cable came about in the early 50s, while real microwave flex cable was approximately 10 years later.</i></p> <p><i>Somewhere along the way it was decided to standardize on a given impedance so that economy and convenience could be brought into the equation. In the US, 50 Ω was chosen as a compromise. There was a group known as JAN, which stood for Joint Army and Navy who took on these matters. They later became DESC, for Defense Electronic Supply Center, where the MIL specs evolved. Europe chose 60 Ω. In reality, in the U.S., since most of the "tubes" were actually existing materials consisting of standard rods and water pipes, 51.5 Ω was quite common. It was amazing to see and use adapter/converters to go from 50 to 51.5 Ω. Eventually, 50 won out, and special tubing was created (or maybe the plumbers allowed their pipes to change dimension slightly).</i></p> <p><i>Further along, the Europeans were forced to change because of the influence of companies such as Hewlett-Packard which dominated the world scene. 75 Ω is the telecommunications standard, because in a dielectric filled line, somewhere around 77 Ω gives the lowest loss. (Cable TV) 93 Ω is still used for short runs such as the connection between computers and their monitors because of low capacitance per foot which would reduce the loading on circuits and allow longer cable runs.</i></p>

<p>At what speeds does this [the impedance mismatch] become more and more critical? I always match source and load characteristic impedance on the PCB, but never specify the cable impedance for custom applications. An example would be CAN [Controller Area Network bus], which specifies 120 Ohm impedance but does not have commercially available hardware-standards. Also, our custom cable house cannot build impedance-controlled cables.</p>	<p>Because Z_0 mismatch can cause problems in so many ways, the question of speed is usually related to attenuation. Every transmission line is a low pass filter and every serial channel can be thought of as a square clock. Every square wave is made up of specific frequencies (Fundamental, 3x, 5x, 7x and so on) in decreasing signal levels. So the solution is more concerned with “when is the frequency so high that attenuation is too great”.</p> <p>When the bandwidth of your cable is limited, the result is too great of attenuation of those higher order harmonics. This is what causes the rounding of the transitions. The goal of having a transmission line impedance matched to the source and load is that the loss associated with the signal propagation is limited to the resistance loss.</p> <p>If your cable company can’t create a cable with 120-ohm Z_0, then you can have tremendous losses per unit length thereby severely limiting the usable length of the cable.</p> <p>There are other issues involved that we might be able to help you with to find a viable solution. Feel free to contact us.</p>
<p>How do you recommend matching a digital HS I2C data line to a transmission line to prevent radiating AM noise that may interfere with a nearby AM receiver?</p>	<p>You are going to have to look at the input impedance of the receiving device. If you match the transmission line impedance to the load, you will minimize the interference of the AM receiver.</p> <p>Note that due to the low speed of the I2C bus (400kHz), no standard impedance is given in the standard.</p>
<p>Can you use power a plane as reference plane for differential line? What are the rules?</p>	<p>You have to use stitching capacitors if you want to use power plane as a reference plane for the differential lines.</p> <p>Stitching capacitors are placed at the transition where the signal lines need to change reference planes. They are typically lower in capacitance as their primary goal is coupling the very highest frequencies between planes.</p>
<p>How much effect do reflections have on logic circuits or "slow" baud rate lines?</p>	<p>Some but not usually very significant unless the mismatch is significant.</p>

<p>Do you match impedance of the trace(s) only to the load, or do you have to take source impedance into account?</p>	<p>Usually the load impedance is matched to the source because of the standards that exist for the devices that both drive and receive specific signals. Therefore, if the transmission line is matched it basically becomes insignificant to the power loss.</p>
<p>How you match impedances at vias?</p>	<p>There are two ways: Approximation or 3D field solver.</p> <p>For an approximation, a via has to be viewed as a series inductor with shunt capacitance. The simpler the shape the less discrete sections are needed to model it. The inductance is fairly straight forward as you just use an equation for a round conductor. The shunt capacitance is much more complicated because you have to deal with so many causes of fringing due to ground pours.</p> <div data-bbox="511 846 868 1314" data-label="Image"> </div> <p>From the picture you can see that when a via is not set up properly, there can be pads around the via on inner layers. These pads cause a non-uniform E-field between the via tube and both the ground vias/pour as well as the other signal via.</p> <p>These pads also impact the inductive characteristics.</p> <p>The best solution would be to remove the pads and create as uniform as possible the physical relationship between the via and the ground. You could change the spacing of the ground pour to keep the capacitance consistent as the wave propagates.</p>
<p>How was that reflection on the vias solved?</p>	<p>By changing the capacitance by way of spacing adjustment between vias and the inductor by way of the hole diameter.</p>
<p>Do you use tear drops to minimize the impedance difference from the track to the via? Does it help?</p>	<p>When dealing with any transitions of microwave structures, abrupt transitions are a no-no. For example, if a 50 ohm microstrip is 30 mils wide and terminates into a SMT device with a pad of only 10 mils, there is a taper of the microstrip that is implemented. Any abrupt changes (sharp edges) will cause a reflection. It is just a matter of “at what frequency” does it become problematic.</p>

Modern High-Speed materials have differing dielectric constants (prepreg v. core). How does this impact the impedance of a via that traverses layers with differing dielectric constants?

The greater the ϵ_r , the more capacitance can be created between a via's conductive surfaces and any surrounding grounds. Therefore, you are creating different shunt capacitors through each material type.



Because of the different dielectric constants of the materials in the stackup shown, there would be different capacitances between the via and any reference plane. E. g. C12 would not be the same as C23.

This stackup creates 3 different shunt capacitances. And, specifically because the dielectric constants are different, then each capacitor is that much more complicated. Furthermore, the impact any ground pour will have is very difficult to calculate. This is why at the upper boundaries of SERDES designs we recommend 3D EM simulations.