FAQ #1 for Spring 2009 Newsletter

Why is the PCI bus impedance specification 65 ohms?

Many times the question arises in my classes or in discussions in groups such as the SI reflector on the Internet. Nearly all other logic protocols specify a 50 ohm impedance for the signal traces and terminations of single ended signals. Why, then, is the PCI bus specification 65 ohms and does it really need to be 65 ohms? Requiring both 50 ohm and 65 ohm impedances on the same PCB usually involves some very difficult layout and PCB stackups to accommodate two such different impedances.

The impedance requirement comes from the PC industry where the PCI bus was first used. Initially, clock speeds and rise times were so slow that impedance did not matter. As a result, PCB layout rules were put in place that optimized PCB yield at the lowest possible cost. PCB motherboards on which the PCI bus is used were 4 layers stacked as follows:

Top- Signal layer 5 mil traces, 5 mil spaces
Dielectric layer- 5 mils, er about 4.2
Layer 2- plane layer, either ground or Vdd
Dielectric core layer- 42 mils, er about 4.5
Layer 3- plane layer, either ground or Vdd
Dielectric layer- 5 mils, er about 4.2
Bottom- Signal layer, 5 mil traces, 5 mil spaces

This resulted in a PCB about 63 mils thick. Why 63 mils? This is a bit like the reason for the spacing of railroad rails. (The width of Roman chariot wheels is alleged to be the origin of this spacing!) Early PCBs were made from Bakelite, a material used for the top ply of a piece of plywood manufactured for work benches. One ply was 1/16 of an inch or about 63 mils. When edge connectors were needed for this material, they were specified at 63 mils to fit this material. It has been decades since Bakelite was used, but we still build PC motherboards 63 mils thick.

As speeds of processors increased and the rise and fall time of signals got ever faster, reflections began to cause failures. Reflections are controlled using termination resistors that match the drivers to the transmission lines either as series or parallel terminations. In order to calculate the proper terminator value it is necessary to know the transmission line impedance. Someone measured the impedance of the surface microstrip transmission lines on typical PC motherboards being manufactured using the above stackup with the lines and spaces then in use. It happened by accident to be very near 65 ohms, hence the 65 ohm impedance specification for the PCI bus on a PC motherboard.

As long as designs can be routed in 4 layers as is done with a PCB motherboard, it is not difficult to achieve a 65 ohm impedance. However, many designs incorporate the PCI bus is large designs that contain hundreds or thousands of nets. Making room for such a large number of nets requires many signal layers, most of which must be buried microstrip or stripline layers. It is very difficult to create buried microstrip and stripline layers that are 65 ohms without creating a PCB that is unrealistically thick.

Based on the difficulty creating a high layer count PCB with signal layers that are 65 ohms, a reasonable question to ask is does the PCI bus really need to be 65 ohms to function properly? One way to answer this is to simulate a PCI net using an IBIS model of the PCI driver with 50 ohms and 65 ohms to see what effect the lower impedance has on signal quality. Figure 1 is such a simulation.

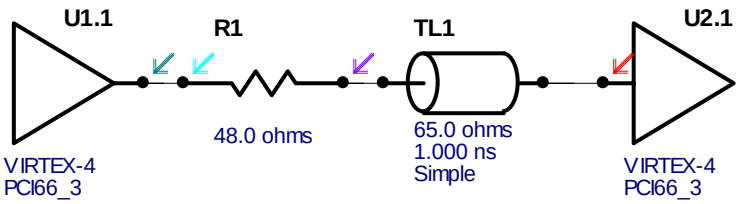

Figure 1. PCI Signal Driving a Six Inch Long Transmission Line

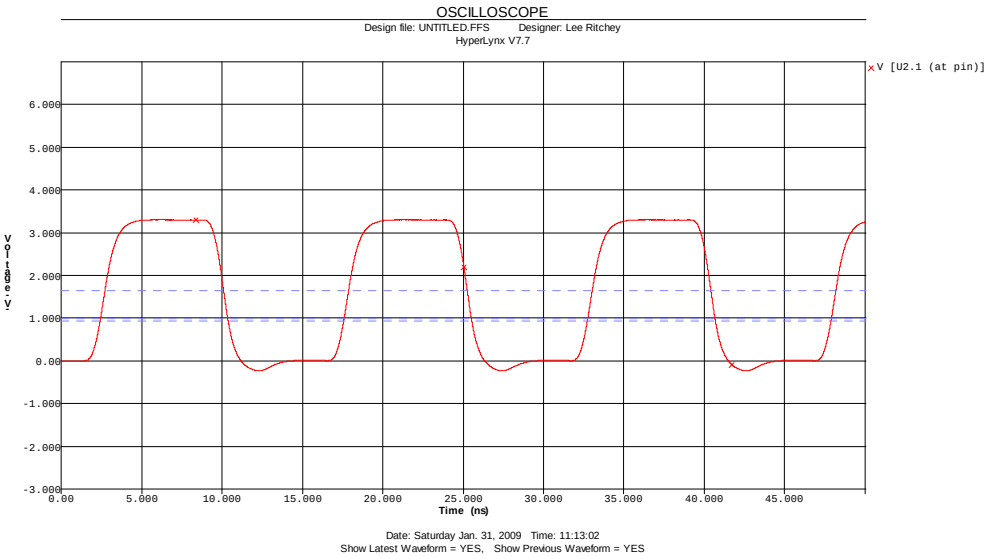

Figure 2. PCI Bus Waveform Driving 50 Ohm Line

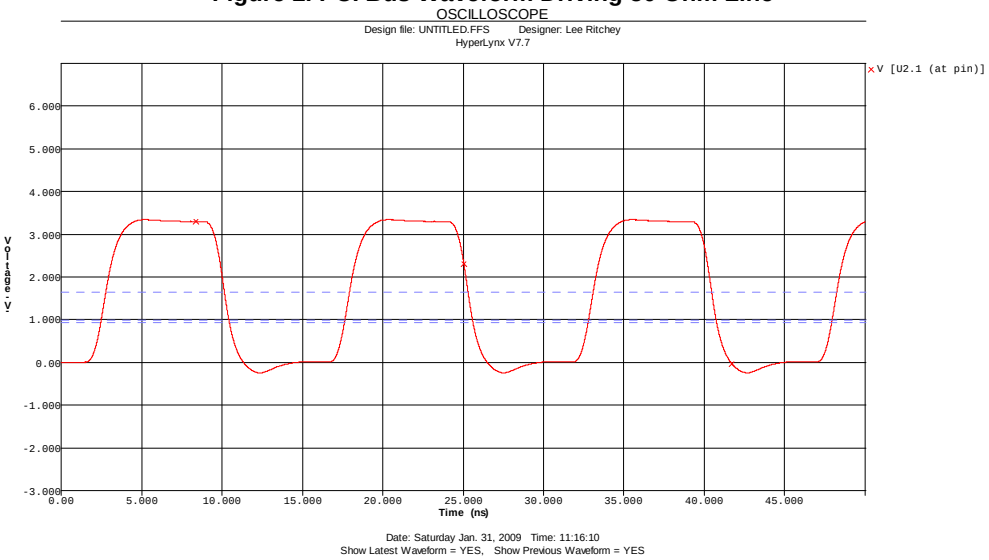

Figure 3. PCI Bus Waveform Driving a 65 Ohm Line

As can be seen by comparing the two waveforms, either 50 ohm or 65 ohm impedance transmission line impedances result in acceptable waveform quality. From this is fair to conclude that it is not necessary to design PCBs with 65 ohm impedance in order for a PCI bus to operate satisfactorily.

Why, then, do so many PCI bus specifications require transmission lines of 65 ohm impedance? The only answer that I have been able to come up with is that it is in the specification, so it must be complied with. Clearly, the bus will work correctly with a 50 ohm impedance removing the need to design PCB stackup for two different impedances at the same time.