



WP323 (v1.0) March 28, 2008

Signal Integrity: Tips and Tricks

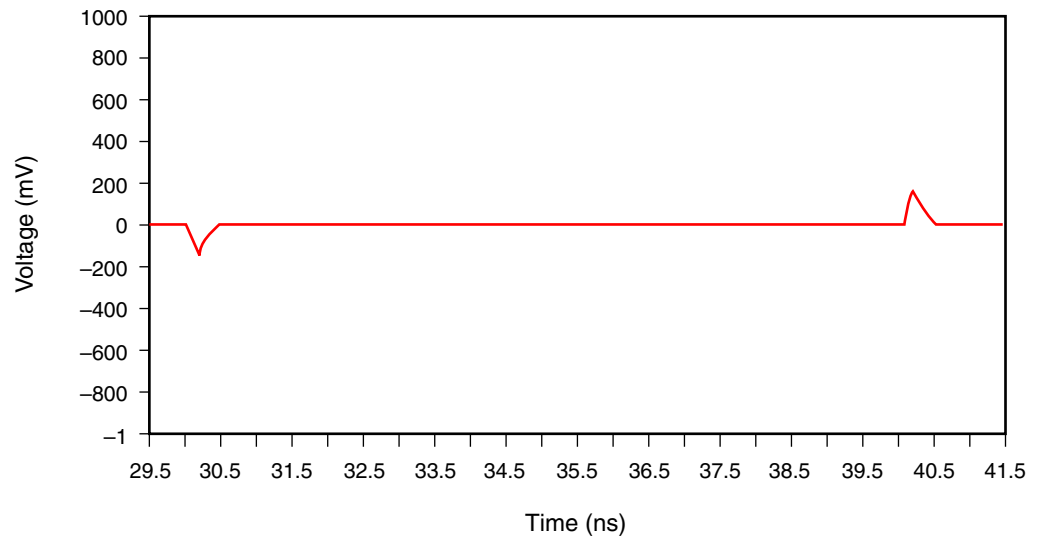
By: Austin Lesea

Signal integrity (SI) engineering has become a necessary requirement for today's high-speed logic signals. It is now critical to gain control of crosstalk, ground bounce, ringing, noise margins, impedance matching, and decoupling (all of which relate to signal integrity) to have a successful design.

Signal integrity describes the environment in which signals must exist. It covers the various techniques and design issues that ensure that signals are undistorted and do not cause problems to themselves, to other components in the system, or to other systems nearby. This white paper describes the theoretical background behind signal integrity and discusses some SI engineering techniques.

The Basics of Signal Integrity

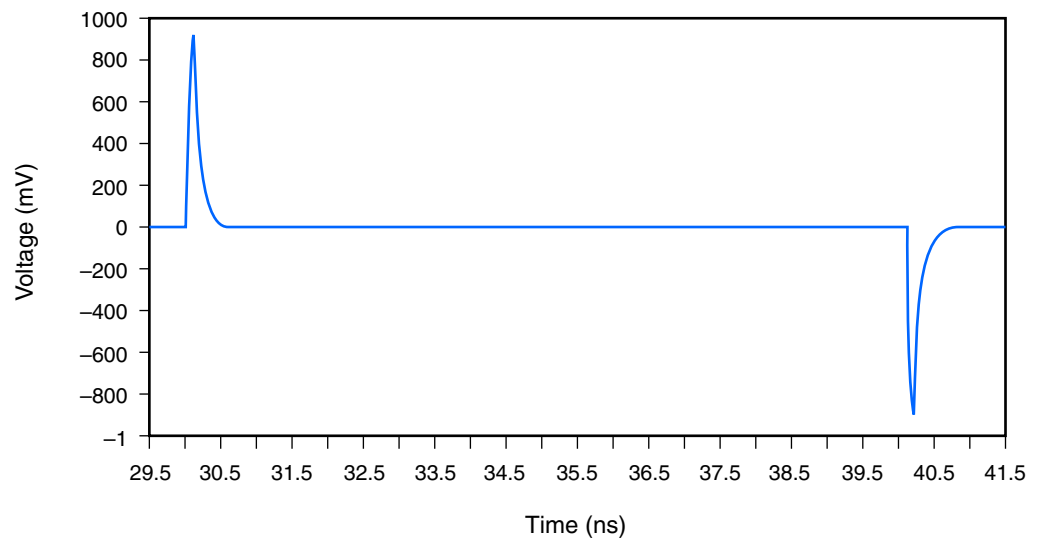
An electrical current in a loop generates a magnetic field. If this magnetic field is changing, it can either radiate energy by launching radio frequency waves or couple to adjacent loops, which is known as inductive crosstalk. [Figure 1](#) shows an example of inductive coupling from one wire to another, illustrating the voltage induced or coupled into the second wire from the first.



WP323_01_011708

Figure 1: Inductive Coupling

The voltage on a line creates an electric field. If this electric field is changing, it can radiate radio waves, or couple capacitively to adjacent lines, which is known as electrostatic crosstalk. [Figure 2](#) illustrates the resulting voltage at a node when it has been interfered with from another node through capacitive coupling.



WP323_02_011708

Figure 2: Capacitive Coupling

There could be both inductive and capacitive coupling present; however, one or the other would probably be dominant and recognizable by the response.

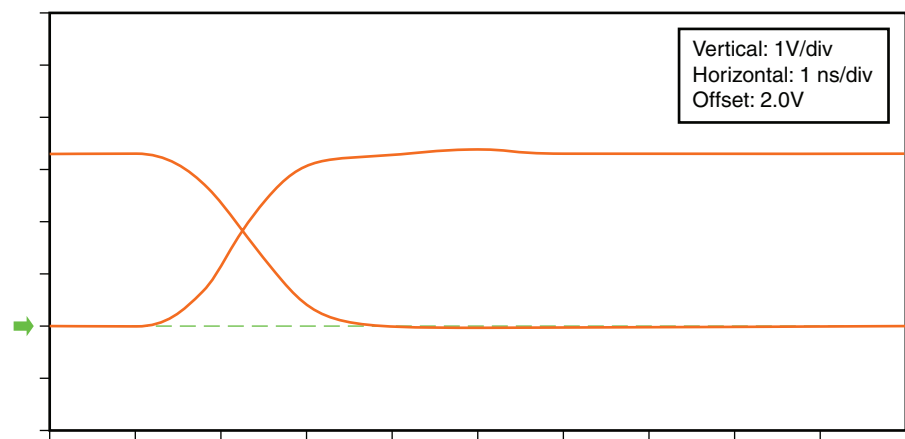
When does a line or loop radiate radio frequency waves? If the line or loop has signal energy at a frequency high enough that the line or loop represents at least a tenth of a wavelength, there is a measurable amount of electromagnetic radiation or radio waves.

Why Antennas Radiate

The impedance of free space is 377Ω . A perfectly matched antenna can be thought of as a transformer that matches the impedance of a line to the impedance of space. Wires go into the antenna and radio waves come out of it. Power launched into the wires appears to go into a resistor. The voltage is in phase with the current and no power is reflected back.

A typical printed circuit board trace over ground is a transmission line with an impedance between 35Ω and 100Ω (depending on the board dimensions and the insulator material). If the driving circuit has low impedance (lower than the line) and the receiving circuit has high impedance (much higher than the line), the line is mismatched. Energy launched from the driver travels down to the end of the line and is reflected back to the driver. At the driver, it is again reflected back to the receiver. This process continues until all of the energy is lost in heat or through radiation into space. If the line is matched and the energy therefore does not bounce back and forth, radiation does not occur because the energy is completely absorbed in the matching resistance. In summary, matched lines do not radiate.

Figure 3 shows an almost perfectly matched system in which the rising and falling edges (superimposed) show little distortion from any mismatch in impedances. These waveforms are characteristic of a driver that is perfectly matched to its load.



WP323_03_120307

Figure 3: Perfectly Matched System

Figure 4 shows the oscilloscope trace for a resistor that was used to series match to an infinite impedance (an open end of a transmission line). The trace shows the voltage across the resistor for a rising and falling edge (superimposed). The power lost in the resistor is the voltage squared, divided by the resistance.

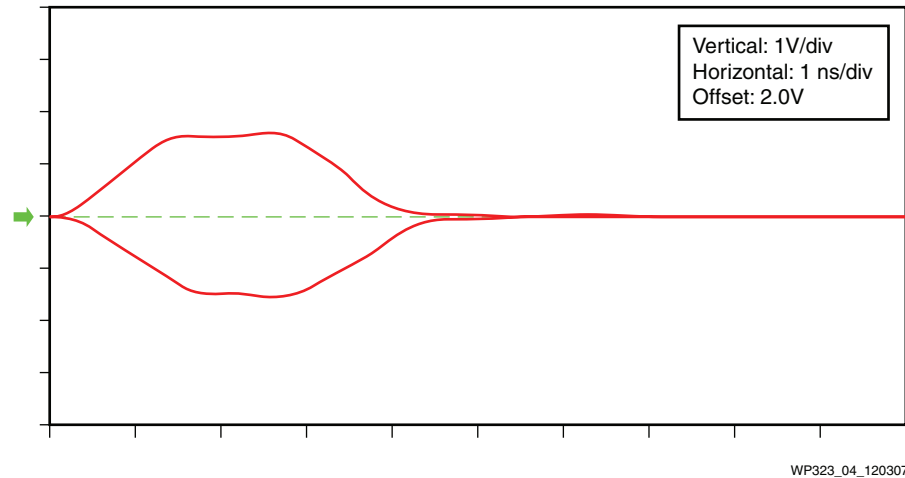


Figure 4: Series Match to Infinite Impedance

Why Lines and Loops Couple

If two lines are placed near to each other, they will couple magnetically, electrically, or both because their electric and magnetic fields overlap in the near field and affect each other. The resulting crosstalk causes delay variation in the transitions or bad signal states altogether.

The solution to this problem is distance or the addition of a conductor between the two lines to constrain the electric field. Further electrostatic shielding would not be worthwhile because there is no good means of separating magnetic fields from one another besides adequate distance. Thus, control of physical dimensions is absolutely necessary to control crosstalk and keep the impedance matched.

How Does Ground Bounce Occur?

If a number of outputs are all capable of driving in the same direction at the same time (such as an address bus that goes from all zeros to all ones), the resulting current from switching all of the lines simultaneously can share a common ground pin. That ground pin can rise significantly above (or below) ground due to the resultant IR drop and the self-inductance of the pin. To prevent the resultant ground potential change from affecting other circuits nearby or other circuits on the same integrated circuit, it is mandatory to provide sufficient ground and power connections and their associated bypass capacitors. Figure 5 shows the ground bounce at 25 °C from 32 simultaneous switching outputs (SSOs) at 3.6V operating at a fast slew rate. The non-switching active-Low output is the ground bounce. Its positive and negative peaks are in the opposite direction of the switching outputs that induced the ground bounce.

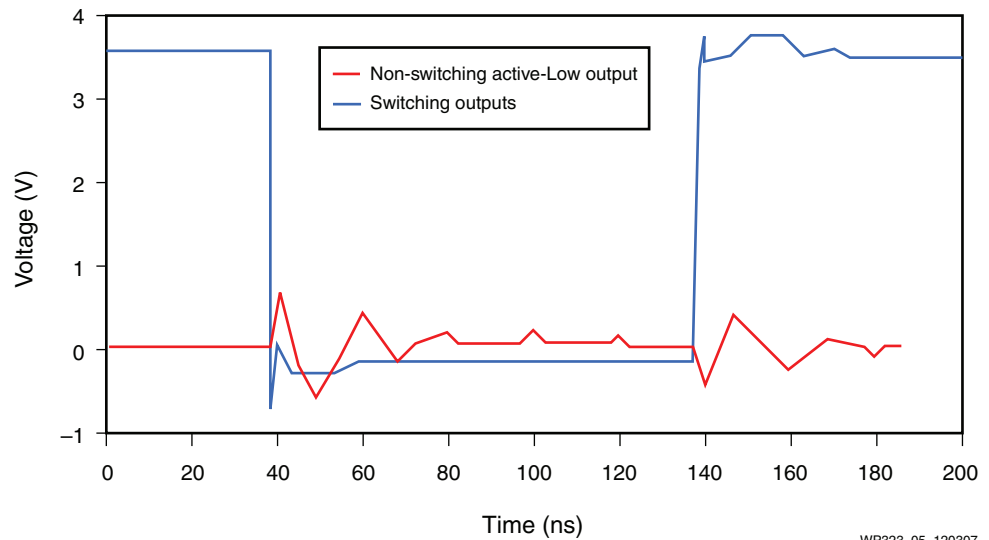


Figure 5: **Ground Bounce**

If a number of input pins are driven and the lines are not matched, significant ringing can occur. This ringing at the receiving end can forward bias protection diodes or structures and cause currents to be injected into the substrate (ground) or power rails. To minimize ground bounce, extra ground and supply pins and good bypassing must be provided, the rate of rise and fall of the lines must be controlled, impedance-matching termination must be provided, and the number of lines must be minimized.

FPGAs Help to Solve Signal Integrity Problems

The use of FPGAs represents an opportunity to control the signal integrity, but only if some work is done in advance of the design itself. First, a floor plan of the input and outputs should be created to make sure that the number and location of SSOs is kept within guidelines. Blocks of outputs must be kept physically distant from the critical inputs to reduce crosstalk coupling and ground bounce disturb problems.

Second, unused I/O pins can be employed as virtual ground or supply pins. They can be programmed to drive a one or a zero at the highest current drive strength (using PCI™, GTL, or LVTTL24 buffers) and tied to the power or ground planes of the printed circuit board. These pins function as additional power and ground pins, keeping the ground and power bounce under control. These virtual pins can be bypassed in the same manner as power or ground pins.

Third, all inputs and outputs should be matched to the printed circuit board traces using a signal integrity analysis package. This package should model the drivers and receivers using a modeling format such as the input/output buffer information specification (IBIS). By adding termination resistors where required, the reflections are minimized and absorbed rather than being radiated. In Virtex®-II devices, termination is available on-chip to accomplish this.

Another technique used in FPGAs is to control the output slew rates. If delay is not an issue, SLOW output attributes can be used to reduce crosstalk and bounce. If slew rate control is not possible, the drive strength should be programmed to a lower strength; for example, using LVTTL2 rather than LVTTL12 drivers. The resulting higher output resistance absorbs reflections and improves signal integrity. External resistors are not necessary through the use of certain SelectIO™ interface options.

Why Haven't I Heard of This Before?

Not long ago, the highest frequencies encountered in digital circuits were below 20 MHz. To be a transmission line or an antenna at these frequencies, a structure had to be about one meter (3 feet) long. Practical circuits did not behave like transmission lines or antennas.

Now that clock rates are at 320 MHz and edge rates are a few hundred picoseconds, interconnects as short as 25 mm (1 inch) begin to look like transmission lines and antennas. Digital systems engineers are only now becoming aware of signal integrity issues, but those involved in microwave radio systems design have used signal integrity engineering for more than 40 years.

Additional Resources

- Johnson, Howard, and Martin Graham. 1993. *High Speed Digital Design: A Handbook of Black Magic*. Prentice Hall.

Revision History

The following table shows the revision history for this document:

Date	Version	Description of Revisions
03/28/08	1.0	Initial Xilinx release. Based on a previously published <i>Tech Xclusive</i> article by the same author.

Notice of Disclaimer

The information disclosed to you hereunder (the "Information") is provided "AS-IS" with no warranty of any kind, express or implied. Xilinx does not assume any liability arising from your use of the Information. You are responsible for obtaining any rights you may require for your use of this Information. Xilinx reserves the right to make changes, at any time, to the Information without notice and at its sole discretion. Xilinx assumes no obligation to correct any errors contained in the Information or to advise you of any corrections or updates. Xilinx expressly disclaims any liability in connection with technical support or assistance that may be provided to you in connection with the Information. XILINX MAKES NO OTHER WARRANTIES, WHETHER EXPRESS, IMPLIED, OR STATUTORY, REGARDING THE INFORMATION, INCLUDING ANY WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, OR NONINFRINGEMENT OF THIRD-PARTY RIGHTS.