Significant of Signal Integrity in High Speed Board Design

Kamal B. Parmar[1] Paresh R. Valand[2]

[1, 2] vlsi And Embedded System (E.C.)

[1, 2] U.V.Patel College Of Engineering,Ganpat University. Kherva, Gujarat, India.

Abstract--- Digital designs are partial without transmission lines. It has no usually suffered by issues related with transmission line effects. At lower Frequencies the signals remain within data description and the output data same as input data. But as system speeds increase, the higher frequency impact on the system output means that not only the digital properties, but also the analog effects within the system are not ignorable. These problems are likely to come to the forefront with increasing data rates for both I/O interfaces and memory interfaces. Transmission line effects can have a considerable effect on the data being sent. At low speeds, the frequency response has little influence on the signal, unless the transmission medium is particularly long. However, as speed increases, high-frequency effects take over and even the shortest lines can suffer from problems such as ringing, crosstalk, reflections, and ground bounce, seriously hampering the response of the signal. How you can overcome these issues that described in this manuscript for good design techniques and simple layout guidelines.

Keyword :- signal integrity, layout guidelines, high speed board design.

I. INTRODUCTION

[1]Signal integrity, as it applies to high speed digital circuits and systems, is often poorly defined. As a result, the effects that the term attempts to describe may not be completely understood by the engineers designing and testing those systems. In this article, we will try to identify the behaviors that collectively can be defined as *signal integrity*.

The word integrity is defined as: "wholeness; completeness; having unimpaired action." Thus, signal integrity deals with the factors that cause a *deviation* from an unimpaired digital signal. Signal integrity is usually described as relating mainly to the effects of physical structures on ideal digital signals, but other effects are also involved, such as clock signal timing, power distribution and noise. The magnitude of the effects of physical structures is frequency dependent, increasing with higher operating frequencies. Signal integrity is a relatively recent concern, becoming a critical element in digital design as clock speeds have increased. Digital signals must now be handled like microwave signals, since the bandwidth required to carry those digital signals extends well into the microwave frequency range.

Likewise, high speed reduces the margin of error for clock timing signals. Accurately distributing the system clock to multiple circuits on a high speed board requires an unprecedented level of precision. At high speeds, timing errors are magnified, both in timing pulse alignment and in rise/fall times that can cause overlapping signals or violations of digital circuit setup and propagation delay times.

II. CAUSES OF SIGNAL INTEGRITY

Timing is everything in a high-speed system. Signal timing depends on the delay caused by the physical length that the signal must propagate. It also depends on the shape of the waveform when the threshold is reached. Signal waveform distortions can be caused by different mechanisms. But there are three mostly concerned noise problems:

A. Reflection Noise:

Due to impedance mismatch, stubs, vias and other interconnect discontinuities.

B. CrosstalkNoise:

Due to electromagnetic coupling between signal traces andvias. Power/Ground Noise Due to parasitic of the power/ground delivery system during drivers' simultaneous switching output (SSO). It is sometimes also called Ground Noise.

III. CORE ISSUES OF SIGNAL INTEGRITY

The various causes of signal integrity problems into related areas.

1. Transmission line effects - losses and reflections in interconnecting traces, including package leads, vias and connectors, as well as impedance matching to the active devices.
2. Coupling effects - crosstalk between signal lines, or between signal and clock lines.
3. Ground currents - Signal return currents (the "other side" of transmission lines); propagated signal, clock and noise; plus DC "ground bounce."
4. Power integrity - DC supply distribution and decoupling; plus unwanted signal or clock propagation through power distribution circuits.
5. Electromagnetic Interference (EMI) - External noise ingress, self-interference, control of radiated emissions.
6. Circuit design issues—Clock distribution, timing errors, logic process sequencing.

IV. WHERE SI PROBLEM HAPPEN

Since the signals travel through all kinds of interconnections inside a system, any electrical impact happening at the source end, along the path, or at the receiving end, will have great effects on the signal timing and quality.

V. EFFECT OF SI

When the data stream is evaluated end-to-end, poor signal integrity will result in excessive data errors. Examination of the digital signal waveform with a high speed oscilloscope may show reduced amplitude, droop or tilt; slowed rise/fall times; increased noise floor; and missing pulses or split pulses (missing segments of pulses). Eye diagram analysis will show a closed eye with poor, or non-existent, logic state detection margin.

The next step is identifying the causes of the observed problems. As individual sources of impairment are identified and corrected, smaller effects will become apparent, eventually, all significant problems should be reduced to levels that result in reliable data transmission throughout the system
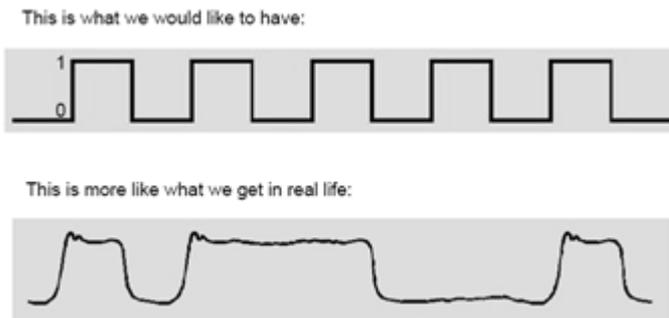
Fig. 1:

"Ignoring signal integrity issues is just asking for trouble"

VI. FUNDAMENTAL PRINCIPLES

Since, we seen they all are types of impairments due to the high frequency effects of physical structures. Classic high frequency effects include:

A. Frequency-dependent behavior –

Capacitance and inductance are frequency dependent. Capacitive reactance decreases with increasing frequency, while inductive reactance increases. Most capacitive effects are from the signal trace to ground, which acting alone would attenuate the signal. Most inductive effects are due to the length of the trace, where increased reactance also affects signal attenuation. Combined capacitance and inductance can also create resonances. At low frequencies, those resonances are usually well above the bandwidth occupied by the digital signal, but with increasing frequency they may directly affect the digital waveform. Typical effects include overshootandringing.

B. Transmission line behavior –

At high frequencies, the length of an interconnection will eventually become an appreciable fraction of a wavelength. Over that length, the applied waveform will undergo transitions in magnitude and phase, defined by the distributed inductance and capacitance of the trace, ground type and dielectric material. Circuit boards are transmission lines, with characteristic impedance established by the above parameters. If the input impedance of a digital device matches the characteristic impedance of the transmission line, all energy will be absorbed by the device, and none will be reflected back toward the source. Thus, the impedance of the high speed board trace is important, as is the terminating impedance of the device. Reflected energy resulting from a mismatched condition will cause signal degradation. Note that those reflections are not caused only by the termination; irregularities in the transmission line can create localized variations in its characteristic impedance, referred to as discontinuities. These may include bends, transitions to vias, thickened areas where components are soldered, and any other physical deviation from a uniform line structure. And as frequency increases, the magnitude of the deviation increases.

C. Electromagnetic effects –

with increasing frequency, we have the shorter wavelengths and increased AC reactance' and the increasingly rapid fluctuation of high frequency signals means that they contain more energy. When that energy is high enough, we can discern its effects, which include coupling, radiation and surface waves.

VII. DISCOVERY SIGNAL INTEGRITY PROBLEM

Direct signal observations and measurements are the only ways to discover many causes of signal integrity-related problems. As always, using the right tool will simplify any task. Most signal integrity measurements are made with the familiar combination of instruments found in most electronics engineering labs: the logic analyzer, the oscilloscope and, in some cases, the spectrum analyzer. Probes and application software (Allegro PCB SI) – to perform tasks like jitter analysis – round out the basic toolkit.

A. Digital Oscilloscope - signal integrity measurement solution is the digitizing oscilloscope. The oscilloscope is used to isolate analog problems once they have been captured, in their digital form, by the logic analyzer. The oscilloscope can display waveform details, edges and noise; it can detect and display transients and it can precisely measure timing relationships like setup and hold times. Since digital errors are often related to analog signal integrity problems, the oscilloscope is a valuable tool in determining the cause of a digital fault.
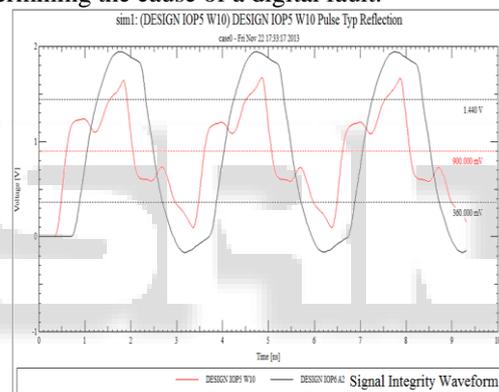

Fig. 2:

B. Allegro PCB SI Tool(Software) - Integrated with Cadence® Allegro® PCB design, editing, and routing technologies, Allegro PCB SI provides advanced signal integrity (SI) analysis both pre- and post-layout. Operating early in the design cycle allows for "what-if" scenario exploration, sets more accurate design constraints, and reduces design iterations. Allegro PCB SI reads and writes directly to the Allegro PCB Editor database for fast and accurate integration of results. It provides a SPICE-based
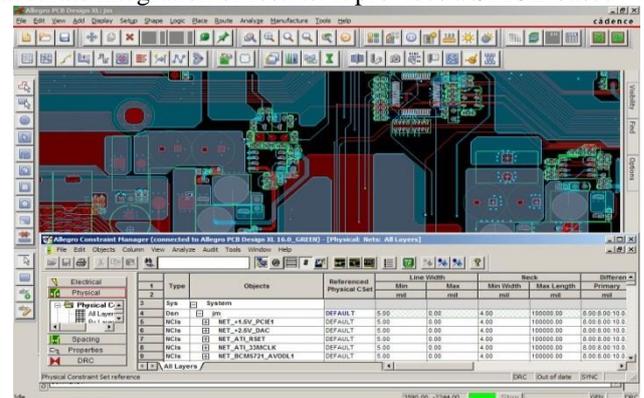

Fig. 3:

simulator and an embedded field solver, and it supports behavioral modeling with a robust modeling language. Bus architecture can be explored pre-layout to compare alternatives, or post-layout for a comprehensive analysis of all associated signals. The Allegro PCB Power Delivery Network (PDN) Analysis Option provides modeling of all power distribution characteristics.

VIII. SETTING UP SIGNAL INTEGRITY PROBLEM

If we are to avoid signal integrity problems in high speed digital circuits, all of the above characteristics must be considered. Today, electromagnetic analysis is necessary. EM analysis is remarkably accurate, and there are a wide range of software tools available. Unfortunately, the mathematics of EM analysis is complex, requiring manipulation of large matrices and large-scale integrations. These calculations are then repeated over many discrete observation locations, with many frequency steps. Even with powerful computers and multi-core techniques, large problems cannot be solved quickly. So we prefer to use that powerful design tools for EM analysis.

An IC package or PCB designer removes signal integrity problems through these techniques:

1. Placing a solid reference plane adjacent to the signal traces to control crosstalk.
2. Controlling the trace width spacing to the reference plane to create consistent trace impedance.
3. Using terminations to control ringing.
4. Route traces perpendicular on adjacent layers to reduce crosstalk.
5. Increasing spacing between traces to reduce crosstalk.
6. Providing sufficient ground (and power) connections to limit ground bounce (this sub discipline of signal integrity is sometimes called out separately as power).
7. Distributing power with solid plane layers to limit power supply noise.
8. Improved clock and data recovery (CDR) circuitry with low jitter/phase noise[3]

IX. SI SOFTWARE VENDOR:

- Ansoft (www.ansoft.com)
- Applied Simulation Technology (www.apsimtech.com)
- Cadence (www.cadence.com)
- Hyperlynx (www.hyperlynx.com)
- Incases (www.incases.com)
- Mentor Graphics (www.mentorg.com)
- Quantic EMC (www.quantic-emc.com)
- Sigrity (www.sigrity.com)
  Viewlogic (www.viewlogic.com)

X. CONCLUSION

Maintaining the integrity of the signal waveform is essential for accurate, reliable operation of high speed digital systems. Signal integrity involves a wide range of behaviors that create impairments to a digital signal waveform. This note focused on the range of high frequency effects common to classic "radio" and "microwave" engineering, and which now must be applied to digital design. Much work has been done to analyze and solve signal integrity issues, but much more is needed in this important area of design.

REFERENCES

[1] Defining Signal integrity: The Characteristics of High Speed Digital Signal By Gary Breed – Editorial Director http://www.highfrequencyelectronics.com/Archives/Apr09
[2] High Speed Digital interconnector and Signal integrity- Sabritec http://www.sabritec.com/technotes/PDF/High_Speed_Digital_Tutorial.pdf.
[3] "Using Clock Jitter Analysis to Reduce BER in Serial Data Applications", Application Note, literature number 5989-5718EN, Agilent Technologies
[4] Rob Kelley, "Choosing the Right Signal Integrity Tool", Electronic Design, pp. 78-80, September,1995.
[5] Lisa Maliniak, "Signal Analysis: A Must For PCB Design Success", Electronic Design, pp. 69 81,September,1995.
[6] High Speed Rules – Part 1, Signal Integrity is Here by Jyrki Keinänen - CHILI devices® International 2006.
[7] "High-Speed Digital Design: A Handbook of Black Magic", Howard W. Johnson and Martin Graham Prentice Hall, 1993.