

Signal Integrity

For PCB Designers



SIERRA
CIRCUITS

Table of Contents

1.	What is Signal Integrity?.....	3
2.	Need for Signal Integrity.....	4
3.	What leads to Signal Integrity issues in a PCB?.....	5
3.1	Impedance discontinuities.....	5
3.2	Reflections, Ringing, Overshoot and Undershoot.....	6
3.3	Crosstalk.....	8
3.4	Via Stub.....	10
3.5	Skew and Jitter.....	10
3.6	Signal Attenuation.....	10
3.7	Ground Bounce.....	11
3.8	Power and Ground Distribution Network.....	11
3.9	EMI Noise.....	11

Signal Integrity for PCB Designers

The moment you hear the term signal integrity, you might start yawning. But once you have an understanding of it, you will be a master in this field for sure.

Let's get straight to the point.

1. What is Signal Integrity?

Signal integrity (SI) signifies the signal's ability to propagate without distortion. Signal integrity is nothing but the quality of the signal passing through a transmission line.

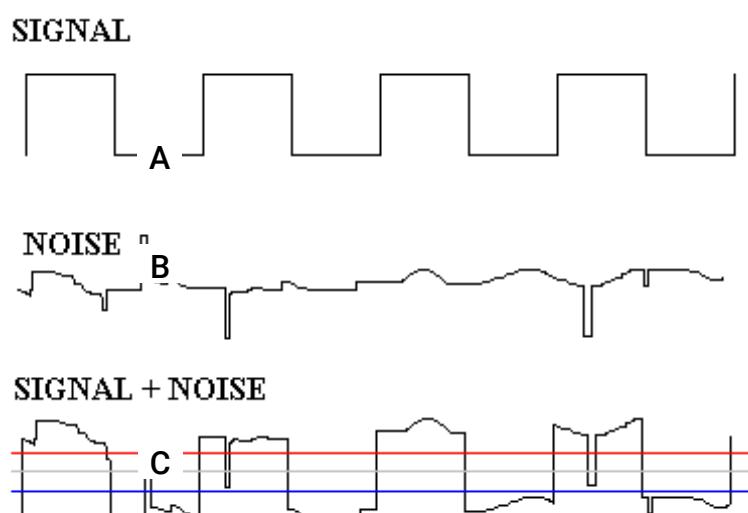
Fundamentally, signal integrity issues must be taken care of during the PCB design phase. Once the PCB has been designed, there is little one can do to improve signal integrity.

A simple analogy for your understanding:

AM signal -> not so clear (distorted signals)

FM signal -> more clear (better signal integrity)

To be more descriptive, signal integrity is the measurement of the quality of an electrical signal typically in electronic printed circuit board. In digital electronics, a stream of binary values is represented by a voltage (or current) waveform. However, digital signals are fundamentally analog in nature, and all signals are subject to effects such as noise, distortion and loss.



Over short distances and at low bit rates, a simple transmitting line can transmit with sufficient fidelity. At high bit rates and over longer distances, transmitting lines can have different effects and degrade the electrical signal to the point where errors occur and the system or device fails.

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 integrity of the signal.

Figure A above shows an ideal signal. Figure C shows the effect of noise on an ideal signal. In the following sections, we discuss the effects of a noisy signal on data integrity. In short, a noisy signal could result in bad data and therefore faulty operations.

2. Need for Signal Integrity

Signal integrity signifies the signal's ability to propagate without distortion.

When we have signal integrity issues in a PCB, it may not work as desired. It may work in an unreliable manner – works sometimes and sometimes not. It may work in the prototype stage, but often fail in volume production; it may work in the lab, but not reliably in the field; it worked in older production lots, but fails in new production lots, etc.

A signal is said to have lost its integrity when:

- It gets distorted, i.e. its shape changes from the desired shape.
- Unwanted electrical noise gets superimposed on the signal, degrading its signal to noise (S/N) ratio.
- It creates unwanted noise for other signals and circuits on the board.

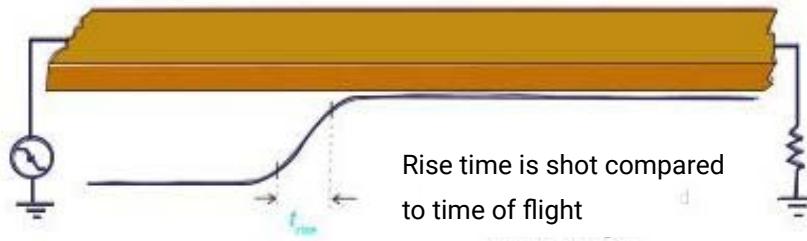
A PCB is said to have requisite signal integrity when:

- All signals within it propagate without distortion.
- Its devices and interconnections are not susceptible to extraneous electrical noise and EMI – electromagnetic interference – from other electrical products in its vicinity as per or better than regulatory standards.
- It does not generate or introduce or radiate EMI in other electrical circuits/cables/products either connected to it or present in its vicinity, as per or better than regulatory standards.

3. What Leads to Signal Integrity Issues in a PCB?

Perhaps the most important cause of signal integrity issues in a PCB is faster signal rise times. When circuits and devices are operating at low-to-moderate frequencies with moderate rise and fall times, signal integrity problems due to PCB design are rarely an issue. However, when we are operating at high (RF & higher) frequencies, with much shorter signal rise times, signal integrity due to PCB design becomes a very big issue.

SIGNAL EDGE MOVING ACROSS A TRACE IN HIGH-SPEED



Factors that contribute to signal integrity degradation:

Generally speaking, fast signal rise times and high-signal frequencies increase signal integrity issues.

For analytical purposes, we can divide various signal integrity issues into the following categories:

3.1 Impedance discontinuities

As we mentioned earlier, if the signal encounters a discontinuity in impedance during its travel, it will suffer reflections which cause ringing and signal distortion. Discontinuities in the line's impedance will occur at the point of encountering one of the following situations:

- When a signal encounters a via in its path.
- When a signal branches out into two or more lines.
- When a signal return path plane encounters a discontinuity, like a split.
- When the line stubs are connected and are at a maximum 1/4th the wavelength of the switching speed of the driver.
- When a signal line starts at the source end.
- When a signal line terminates at the receiver end.
- When signal and return paths are connected to connector pins.

And, faster the signal rise time, greater will be the signal distortion caused by impedance discontinuities.

We can minimize signal distortion due to line impedance discontinuities by:

- Minimizing the effects of discontinuities caused by vias and via stubs by using smaller microvias and HDI PCB technology.
- Reducing trace stubs lengths.
- Routing traces in daisy chain fashion rather than multi-drop branches when a signal is used at more than one place.
- Proper terminating resistors at the source.
- Using differential signaling and tightly coupled differential pairs, which are inherently more immune to discontinuities in signal return path planes.

3.2 Reflections, Ringing, Overshoot and Undershoot

What is Reflection?

When a signal is transmitted in a transmission line, some of the signal power may be reflected back to its transmitter rather than being carried all the way along the trace to the far end. Whenever the impedance changes in a circuit, some amount of reflection will happen. The reflected signal will travel back until it encounters another change in impedance and gets reflected again.

Influence of reflection:

- Signal distortion caused by reflection
- Overshooting and undershooting caused by reflection

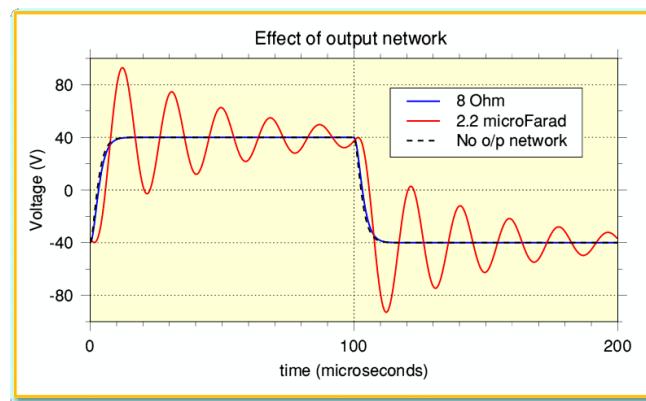
How to reduce reflection noise:

- Maintain the constant impedance
- Maintain good ground grading
- Use series termination resistor and place near the source point. The series termination resistor should be placed within 1/6th wavelength of the switching speed of the driver.

What is Ringing?

Ringing is a voltage or current output that oscillates like a ripple on a pond when it's seen on an oscilloscope. The oscillation is a response to a sudden change in the input signal, like turning it on or switching.

Daniel Beeker and **Rick Hartley** explained, "Ringing is the result of having the driver farther away from the receiver than 1/4th wavelength. This results in a first order reflection of more than the incident wave that returns to the driver and becomes a depletion wave at a lower voltage going back to the receiver, until all of the energy finally either goes into the receiver, is converted to heat in the conductors and dielectric or mostly radiates."



Influence of ringing:

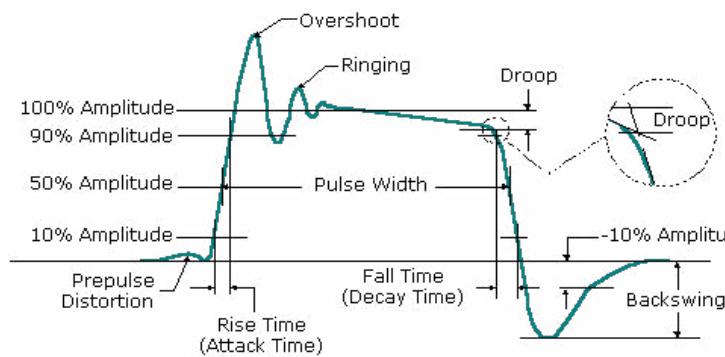
- Increased EMI
- Increased current flow
- Decreased performance
- Audible feedback

What is Overshooting and Undershooting?

A theoretical instantaneous transition of signal allowed maximum upper and lower amplitude.

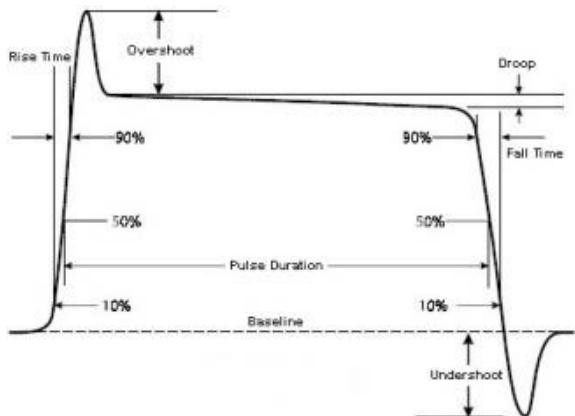
Overshoot:

When the signal transits from lower value to higher value and the value of the transit signal is more than the actual value, then overshoots occur.



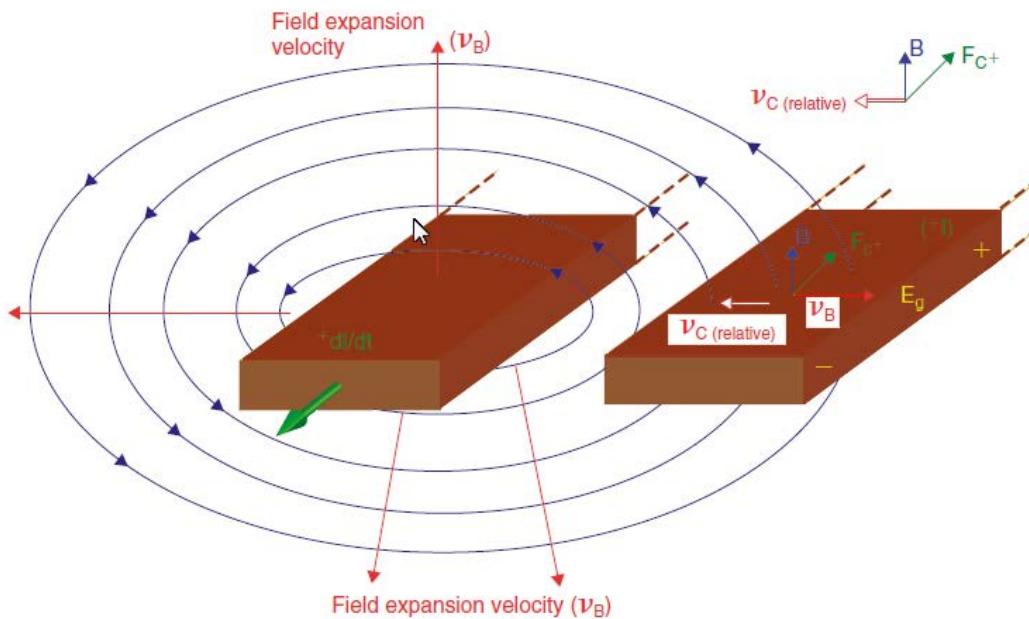
Undershoot:

When the signal transits from higher value to lower value and the value of the transit signal is more than the actual value, then undershoots occur.



3.3 Crosstalk

One signal transmitting in one channel or circuit in a transmitting system creates an undesired effect on another circuit or channel, as shown in the below image.



Crosstalk occurs when there is coupling of energy from aggressor signal to victim signal (typically two tracks close to each other) in terms of the interference of electric and magnetic fields. The electric field is coupled via mutual capacitance between the signals. On the other hand, the magnetic field is coupled via mutual inductance between the signals.

Mutual capacitance:

When two traces run parallel to each other and are separated by a dielectric they behave as parallel plates of a capacitor and when the traces are at two different voltages an electric field is generated between them. Any variation of voltage in one of the traces will induce current in the other trace due to the electric field variation. This capacitance between two traces is called mutual capacitance.

Mutual inductance:

A trace carrying current has a magnetic field around it. If there is another trace close to the first trace carrying current this magnetic field will couple with the second trace. By Faraday's law if there is a variation in current in the first trace the magnetic field will change which will induce a voltage in the second trace. The inductance due to magnetic field coupled traces is called mutual inductance.

Techniques for decreasing crosstalk:

- Increase the spacing between signal lines as much as routing restrictions allow.
- When designing the transmission line, the conductor should be placed as close to the ground plane as possible. This couples the transmission line tightly to the ground plane and decouples it from adjacent signals.
- Implement differential routing techniques where possible.
- To avoid coupling, the signals should be routed on different layers orthogonal to each other.
- Reduce parallel run lengths between signals.

Crosstalk Noise

A fast voltage or current transition on a signal line or return path plane may couple onto adjacent signal lines causing unwanted signals called crosstalk and switching noise on the adjacent signal lines. The coupling occurs due to mutual capacitance and mutual inductance. In uniform transmission lines, a relative amount of capacitive and inductive coupling is comparable. If there are discontinuities in transmission lines, usually inductive coupling dominates, and switching noise results. And as always, faster rise time signals create more crosstalk and switching noise.

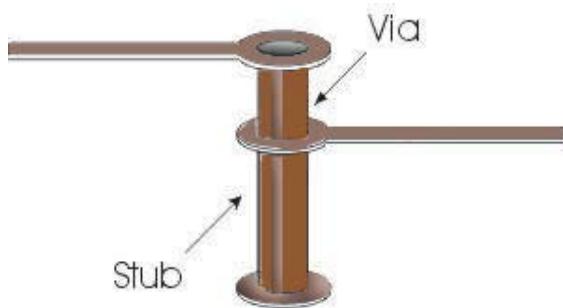
Crosstalk and switching noise can be reduced by:

- Increasing the separation between adjacent signal traces.
- Making the signal return paths as wide as possible, and uniform like uniform planes, and avoiding split return paths.
- Using a lower dielectric constant PCB material.
- Using differential signaling and tightly coupled differential pairs, which are inherently more immune to crosstalk.

3.4 Via Stub

When a routed signal starts from the top layer and ends with some inner layer, the remaining portion form the inner layer to the bottom layer is a via stub.

Daniel Beeker and **Rick Hartley** said, "A stub is a single piece of conductor, and unless there is a pair of vias next to each other - one ground and one signal - or a signal via and a ground plane, the field does not see the stub except as very high impedance."



A via stub acts like a resonant circuit with a specific resonant frequency at which it stores maximum energy within it. If the signal has a significant component at or near that frequency, that component of the signal will be heavily attenuated due to the energy demands of the via stub at its resonant frequency.

3.5 Skew and Jitter

Signals take finite times as they travel on a PCB from source to receiver. The signal delays are proportional directly to signal line lengths and inversely proportional to signal speed on the specific PCB layers. If data signals and clock signals do not match in overall delays, they would arrive at different times for detection at the receiver, and this would cause signal skews; and excessive skew would cause signal sampling errors. As signal speeds become higher, the sampling rates are also higher, and allowable skew gets smaller, causing greater propensity for errors due to skew.

TIP

Skew in a group of signal lines can be minimized by signal delay matching, primarily by trace length matching.

3.6 Signal attenuation

Signals suffer attenuation as they propagate over PCB lines due to losses caused by conducting trace resistances (which increases at higher frequencies due to skin effect) and dielectric material dissipation factor Df. Both these losses increase as frequency increases, therefore higher frequency components of signals will suffer greater attenuation than do the lower frequency components; this causes reduction in signal bandwidth, which then leads to signal distortion by increase in signal rise time; and excessive signal rise time increase results in errors in data detection.

TIP

When signal attenuation is an important consideration, one has to choose the right type of low loss high-speed material and proper control over trace geometries to minimize signal losses.

3.7 Ground Bounce

Ground bounce is a form of noise that occurs during transistor switching i.e., when the PCB ground and the die package ground are at different voltages.

Techniques for decreasing ground bounce:

- Implement decoupling capacitors to local ground.
- Incorporate serially-connected current-limiting resistors.
- Place decoupling capacitors close to the pins.
- Run proper ground.

3.8 Power and Ground Distribution Network

Power and ground rails or paths or planes have very low, but FINITE nonzero impedances. When devices' output signals and internal gates switch states, currents through power and ground rails/paths/planes change, causing a voltage drop in power and ground paths. This will decrease the voltage across the power and ground pins of the devices. Higher the frequency of such instances, and faster the signal transition times, and higher the number of lines switching states simultaneously, greater is the voltage decrease across power and ground rails. This will reduce signals' noise margins, and if excessive, would cause devices to malfunction.

To reduce these effects, the power distribution network has to be so designed as to minimize the power system's impedance:

- Power and ground planes should be placed as close together.
- Multiple low inductance decoupling capacitors should be used across power and ground rails and they should be placed as close to device power and ground pins as possible.
- Use device packages with short leads.

3.9 EMI Noise

EMI increases with frequency and faster signal rise times. Radiation far-field strength increases with frequency linearly for single-ended signal currents, and squarely for differential signal currents.

All the above mentioned design guidelines will help a designer in crafting an efficient PCB design for high-speed circuit boards.

SIERRA CIRCUITS

Sierra Circuits
1108 West Evelyn Avenue
Sunnyvale, CA 94086
+1 (408) 735-7137

www.protoexpress.com

