

A Comprehensive Guide to Signal Integrity Simulation in PCB Design: What You Need To Know



Table of Contents

Introduction *P.1~2*

What is signal integrity? Why is signal integrity simulation important?

What are the different types of signal integrity simulations? P.3

Physical Modeling Transmission Line Modeling

What are the tools and techniques used to simulate signal integrity? P.4

Impedance simulation MoM Simulation EM Field Simulation

How to select the right simulation tool? P.5~6

Capabilities Limitations Design requirements

What are the key steps involved in signal integrity simulation? P.7-8

Define the Stackup Select the right design parameters Run the simulation

What are the common signal integrity issues? P.9

Excessive attenuation Excessive crosstalk Impedance Discontinuities Propagation Mismatches

Conclusion P.10

AUTODESK

A Comprehensive Guide to Signal Integrity Simulation in PCB Design: What You Need To Know



SI Extension in Fusion Electronics showing impedance of a highspeed trace

Are you a PCB designer looking for a comprehensive guide to signal integrity simulation? Look no further! This guide provides a detailed overview of the key aspects of signal integrity simulation in PCB design, from the basics to the more advanced concepts. We'll cover essential topics such as the fundamentals of signal integrity, the importance of signal integrity simulation, the different simulation techniques, and the tools and techniques used to perform signal integrity analysis. This guide will provide you with the knowledge and understanding you need to design successful PCBs with optimal signal integrity.



What is signal integrity?

Signal integrity refers to the ability of a circuit to transfer high-speed signals without any degradation due to impedance mismatches, stray capacitance, inductance, crosstalk, noise, or other factors. It is an important aspect of circuit design, as high-speed circuits operate with frequencies in the hundreds of megahertz or gigahertz, and any impedance mismatch, crosstalk, or stray capacitance can cause significant signal degradation. Signal integrity is a critical part of any circuit design, particularly in high-speed circuits, as the signals being transmitted are very sensitive to any impedance mismatches, stray and parasitic capacitance, inductive coupling or other factors that could lead to signal degradation. High-speed circuits operate with frequencies in the hundreds of megahertz or gigahertz, while lower speed circuits operate with frequencies in the kilohertz or tens of megahertz.

Why is signal integrity simulation important?

Signal integrity simulation is important as it allows you to predict the impact of impedance mismatches, crosstalk, inductance, propagation delays and other factors on a signal's performance before you fabricate your PCB boards. By simulating the performance of your circuit, you can understand the impact of each of these factors and avoid any unexpected issues once your circuit is in operation. Impedance mismatches result in signal degradation and ringing. Propagation delays result in signal losses and data corruption if signals do not arrive in the time they are needed. These factors can have a significant impact on a signal's performance and can even cause your PCB to fail if not properly accounted for.



Screen Capture of SI Extension showing impedance, time delay, resistance, capacitance and inductance of a trace

Signal integrity simulations allow you to optimize your PCB design for performance and minimize the amount of effort required to achieve those results. They allow you to find issues before they make it to the prototype stage, so you can fix them easily with minimal effort. Signal integrity simulations are an excellent tool for optimizing your design.

What are the different types of signal integrity simulations?

While the term "signal integrity simulation" is generally used to refer to the modeling of the circuit to predict performance, there are actually two different types of simulation: Physical Modeling and Transmission Line Modeling.

Physical Modeling

Physical modeling takes into account the components in the circuit (e.g. resistors, inductors, capacitors, and more), the interconnections between components, and how these components affect the signal. It's an end-to-end simulation that will take into account the entire circuit to analyze how the signal propagates through your design.

Transmission Line Modeling

Transmission Line Modeling is a simpler form of signal integrity simulation. It only considers the interconnections between components and doesn't take into account the components themselves. It's a "partial circuit" simulation often used to simulate your PCB trace connections in a circuit.

These two simulation types are similar but have different scopes and complexities. Transmission Line Modeling is a simpler type of simulation that doesn't consider the circuit components themselves. Physical Modeling is an end-to-end simulation that considers all of the circuit components.

What are the tools and techniques used to simulate signal integrity?

There are many tools and techniques used to perform signal integrity simulation. These tools and techniques will depend on your design requirements and the limitations of your PCB design software.

Impedance simulation

Impedance simulation is a popular technique used to perform signal integrity simulation that can be performed with the SI Extension to Fusion Electronics. This extension calculates the impedance of trace sections using a Method of Moments (MoM) technique that factors in the geometry of a trace.

MoM Simulation

The MoM simulation technique is ideal for PCBs as it is fast, accurate and can give you resistance, capacitance, inductance and propagation delays for PCB traces.

EM Field Simulation

If you need to factor in the effect of 3D structures, such as connectors, or need to visualize where electromagnetic (EM) fields are propagating then an EM field simulation is needed. Fusion Electronics allows you to send your to Ansys 3D Field solvers such as HFSS and SIwave for a comprehensive signal integrity analysis including S-parameter extraction, eye diagrams, crosstalk and Simultaneous Switching Noise (SSN).

How to select the right simulation tool?

When selecting a tool or technique to perform signal integrity simulation, you should consider your design requirements, the capabilities of your PCB design tool, and the limitations of your design tool.

Capabilities

First, determine the capabilities of your PCB design tool to determine if it's compatible with the technique you want to use for your signal integrity simulation. Each simulation technique is different and has different requirements, so you'll want to make sure your PCB design tool supports the technique you want to use. The SI Extension for Fusion Electronics is extremely thorough. It provides resistance, impedance, capacitance, and propagation delays based off substrate and copper thickness, ground planes, Dk (dielectric constants) and Df (dissipation factors) of substrates. All these parameters are defined through the layer stack manager making it easy to visualize and enter their values.

ayer	Pairs:								Via Pairs:			Preview:	
#	Name	Material	Туре	Thickness	Weigh	Dk @ 1GH	Df @ 1GH	Condu	Туре	From	То		
	Top SilkScreen	ASP:White	Silk Screen						Thru	1	16	1	
*	Top SolderMask	User	Solder Mask	0.020318		4.30	0.0250					2	
	Top Surface Finish	Lead-Free	Surface	0.02 mm									
1*	Тор	User	Signal	0.035 mm	1oz			580000					
*	Dielectric-1	User	Prepreg	0.210312		4.60	0.0170						
2	Route2	CuFoil:005	Signal	0.018 mm	1/2oz			580000					
*	Dielectric-3	User	Core	1.065 mm		4.60	0.0144						
15	Route15	CuFoil:005	Signal	0.018 mm	1/2oz			580000					
*	Dielectric-5	User	Prepreg	0.210312		4.60	0.0170					15	_
16*	Bottom	User	Signal	0.035 mm	1oz			580000					
	Bottom Surface	Lead-Free	Surface	0.02 mm								16	
*	Bottom SolderMask	User	Solder Mask	0.020318		4.30	0.0250					-	
	Bottom SilkScreen	ASP:White	Silk Screen										

Fusion Electronics Layer Stack Manager showing layers and their properties



Limitations

Next, determine the limitations of your PCB design tool and how they will impact the results of your simulation. Most highspeed PCB design requires trace impedance and propagation delays to meet specification. The MoM simulation technique provided in the SI Extension of Fusion Electronics provides this and does it quickly.

Other simulation techniques have different limitations. For example, S-parameter extraction and full wave EM field simulations will provide a more accurate representation of the propagation of the signal, but it also more expansive in terms of computational resources and requires simulation expertise to setup the models and understand the results.

Design requirements

Finally, determine your design requirements and which technique would be most appropriate for your circuit design. Choosing a simulation that is integrated with your ECAD package makes for fast and convenient simulation.

What are the key steps involved in signal integrity simulation?

To perform signal integrity simulation, there are a number of key steps you'll want to consider. These steps will depend on your design requirements and the technique used for your simulation.

Define the Stackup

First, you'll want to define the PCB stackup as that contains the information needed to perform the simulation calculations. As mentioned earlier, substrate and copper thicknesses along with Dk and Df of substrates are required to perform the calculations used determine impedance. Also, make sure that your simulation factors in trace geometry and vias. Fusion Electronics makes this easy by graphically showing the stackup and allowing stackups to be saved/loaded for future use by others within your organization or from PCB manufacturers.

File	Layer Stack Cleara	nce Distan	ce Sizes	Annular Ring) Sha	pes Supp	ly	
4 Laye	r Stack 🔹		4 5	84 8 <u>x</u>	🕵 🛛 🗹 Edit material propert			
2 Laye	er Stack	1						
4 Laye	er Stack	Material	Туре	Thickness	Weigh	Dk @ 1GH	Df @	
6 Laye	er Stack	ASP:White	Silk Screen	metress	ireigi	DREIM	Die	
10 Lay	ver Stack	User	Solder Mask	0.020318	1oz 1/2oz	4.30 4.60	0.025	
12 Lay	yer Stack	Lead-Free	Surface	0.02 mm				
14 Lay	ver Stack	User	Signal	0.035 mm				
16 Lay	yer Stack	User	Prepreg	0.210312				
Open	Preset File	CuFoil:005	Signal	0.018 mm				
Open From Setup String		User	Core	1.065 mm		4.60	0.014	
15	Route15	CuFoil:005	Signal	0.018 mm	1/2oz			
*	Dielectric-5	User	Prepreg	0.210312		4.60	0.01	
16*	Bottom	User	Signal	0.035 mm	1oz			
	Bottom Surface	Lead-Free	Surface	0.02 mm				

Fusion Electronics can load preset stackup files saved and used by others in your organization or from PCB Manufacturers

Select the right design parameters

Next, you'll want to select the right design parameters for your simulation. This includes the design frequencies, any design constraints, and the materials and widths of the traces and components in your design. Design rules and net classes make it possible to ensure consistency and accuracy in your design as they will ensure your layout adheres to the rules you specify.

Run the simulation

Once you've selected all of the design parameters and techniques for your simulation and laid out some traces, you'll want to run the simulation to generate the results. Depending on the technique you use for your signal integrity simulation, you may want to run multiple simulations for different design scenarios to account for different variations in the design. Your simulation should clearly show anomalies and discontinuities.

What are the common signal integrity issues?

There are many potential issues that can result from signal integrity issues. These issues will depend on the specific components in your design and how each component will affect the propagation of the signal.

Excessive attenuation

Excessive attenuation is when the signal experiences too much attenuation. This can occur if your design has a large amount of capacitance or inductance, which will cause the signal to be significantly attenuated as it propagates through the circuit. PCB substrates with high Df will attenuate high frequencies more than low Df substrates.

Excessive crosstalk

Excessive crosstalk is when the circuit experiences too much crosstalk. This is a common issue that can occur when two circuits are too close together and are experiencing significant crosstalk.

Impedance Discontinuities

Impedance discontinuities are changes in impedance caused by changed in trace geometry or ground planes. When signal experiences a change in impedance, a reflection of part of the signal will occur resulting in ringing and signal degradation and potential Electromagnetic Interference (EMI) issues such as crosstalk.

Propagation Mismatches

Most highspeed signals require that signals appear together at the same time. Propagation delay is proportional to length and is affected by PCB Substrate characteristics and the design of the traces. The SI Extension provided by Fusion Electronics factors this in when calculating delays. Trace meandering is a technique use to lessen the effect of mismatched signal delays.



Conclusion

Signal integrity simulation is an essential design analysis tool that enables engineers to optimize the performance of their circuit and PCB designs. This type of simulation helps engineers to comply with industry standards, avoid costly mistakes, and optimize the design of their PCBs by adjusting trace width, length, thickness, and trace-to-trace spacing. With simulation, engineers can also resolve potential EMI issues by applying ground planes and creating coplanar lines.

Fusion Electronics with the SI extension is an ideal tool for highspeed design PCBs as it gives designer the information they need to comply with modern highspeed communication standards as well as allowing the designer to see the effects changes in substrates have on impedance and signal propagation delays.