AN 958: Board Design Guidelines

Contents

1. Power Distribution Network.....5
1.1. Target Impedance Decoupling Method..... 5
1.1.1. FDTIM Decoupling Concepts..... 5
1.1.2. Determining the Z_{TARGET}..... 6
1.1.3. Determining the f_{TARGET}..... 6
1.1.4. Select Decoupling CAPS to Meet Z_{TARGET}..... 6
1.2. Voltage Regulator Selection.....6
1.3. PDN Design Tool.....7
1.4. Plane Capacitance.....7
1.5. Minimization Parasitic Inductances..... 9
1.5.1. VRM.....9
1.5.2. Decaps.....9
1.5.3. Mounting Inductance.....9
1.5.4. Spreading Inductance.....12
1.5.5. Inter-Plane Capacitance.....12
1.5.6. Conclusion..... 13
1.6. Additional Resources..... 13

2. Gigahertz Channel Design Considerations.....14
2.1. Material Selection and Loss..... 14
2.2. Routing Guidelines for Gigabit Channel Designs.....14
2.2.1. Via Optimization Techniques..... 14
2.3. Minimizing Adverse Effects of Material Glass Fiber Weaves.....18
2.4. Plane Cut-outs Below Surface Mount Pads..... 18
2.5. Transmitter Pre-Emphasis and Receiver Equalization.....18
2.6. Additional Resources..... 19

3. PCB and Stack-Up Design Considerations.....20
3.1. Material Selection and Loss..... 20
3.2. Board Thickness and Vias.....20
3.3. PCB Recommended Layout Footprint Land Pattern..... 20

4. Device Pin-Map, Checklists, and Connection Guidelines..... 21
4.1. High Speed Board Design Advisor..... 21
4.2. Complete Pin Connection Table by Device.....21
4.3. Pin Connection Guidelines By Device..... 21
4.4. Design for Debug with JTAG Pins.....21
4.5. Hot Socketing, POR and Power Sequencing Support.....21
4.6. Implementing OCT.....21
4.7. Unused I/O Pins Guidelines..... 21
4.8. Device Breakout Guidelines..... 22
4.9. Additional Resources..... 22

5. General Board Design Considerations/Guidelines..... 23
5.1. Board Design Process Overview..... 23
5.1.1. Material Selection and Loss..... 24
5.1.2. Cross Talk Minimization.....24
5.1.3. Power Filtering/Distribution..... 25

5.1.4. Unused I/O Pins....27
5.1.5. Signal Trace Routing....28
5.1.6. Ground Bounce....56
5.1.7. Understanding Transmission Lines....60
5.1.8. Impedance Calculation....61
5.1.9. Coplanar Wave Guides....65
5.1.10. Simultaneous Switching Noise Guidelines....67
5.2. Design Security Features in FPGAs....68
5.3. EMI....68
5.4. Discrete Component Selection for High-Speed Design....69
5.4.1. Resistors and Capacitors....69
5.4.2. Inductors and Ferrite Beads....69
5.4.3. SMA Connectors....70
5.5. S-Parameters....71
5.6. Smith Chart....73
5.7. AC Versus DC Coupling....73
5.8. Additional Resources....75

6. Memory Interfacing Guidelines....76
6.1. DDR3 Board Design Guidelines....76
6.2. DDR2 Board Design Guidelines....76
6.3. DDR Board Design Guidelines....76
6.4. QDR II and QDR II+....76
6.5. RLDRAM II....76
6.6. Additional Resources....77

7. Power Dissipation and Thermal Management....78
7.1. Power Management Guidelines....78
7.2. Heat Sinking and Thermal Managment for FPGAs....78
7.3. Additional Resources....78

8. Tools, Models, and Libraries....79
8.1. Design Tools....79
8.2. I/O Buffer Models....79
8.3. Device Libraries....79
8.4. Net Length Reports....79
8.5. Thermal Models....80

9. Reference Designs and Development Kits....81
9.1. Development Kits....81
9.2. Development Kits by Technology....82
9.2.1. DDR3 Memory Technology....82
9.2.2. DDR2 Memory Technology....82
9.2.3. DDR Memory Technology....82
9.2.4. QDR2 SRAM Memory Technology....82
9.2.5. SRAM Memory Technology....82
9.2.6. RLDRAM II Memory Technology....83
9.2.7. FLASH Memory Technology....83
9.2.8. Gigabit Transceiver Technology....83
9.2.9. PCI Express Technology....83
9.2.10. PCI Technology....83
9.2.11. 10/100/1000 Ethernet Technology....83

9.2.12. USB 2.0 Technology.....84
10. Document Revision History for AN 958: Board Design Guidelines..... 85

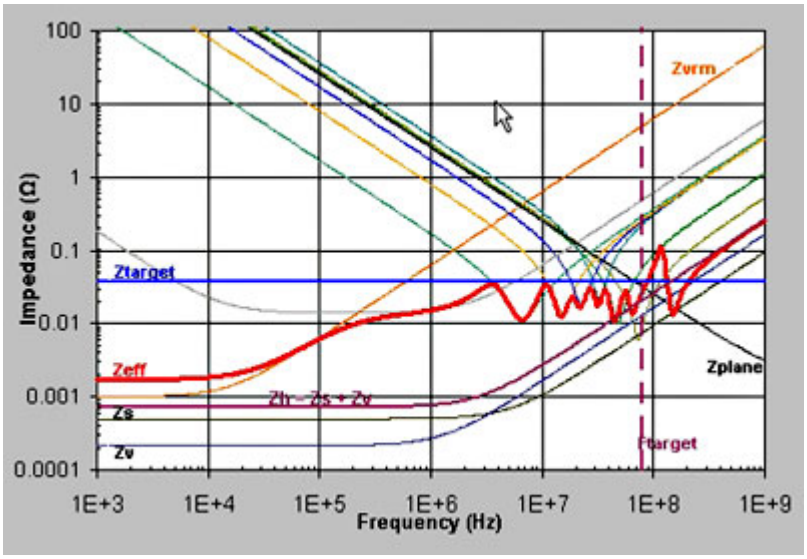
1. Power Distribution Network

1.1. Target Impedance Decoupling Method

An efficient strategy for determining the PCB decoupling scheme is to use the Frequency Domain Target Impedance Method (FDTIM). This method is used by the PDN Decoupling Calculator Tool and is the recommended method for determining the board-level decoupling requirements using Intel FPGAs.

1.1.1. FDTIM Decoupling Concepts

The key concept of the FDTIM method is the determination of the target impedance (ZTARGET) for the power rail under consideration. A reliable decoupling strategy warrants for the effective power rail impedance (ZEFF) to be lower than the ZTARGET from a few KHz up to the highest frequency of interest (fTARGET). #vvh1628537152433/fig_N10015_N1000E_N10001 on page 5 illustrates this concept, where the solid horizontal blue line is the ZTARGET and the dotted vertical brown line is the fTARGET. The solid red ZEFF line has been designed by using various decoupling and bulk capacitors such that its impedance remains below the ZTARGET from DC up to fTARGET. With this design, power integrity is achieved from DC up to the target frequency of decoupling.



To design a reliable decoupling scheme using the FDTIM, perform the following calculations:

1.1.2. Determining the Z_{TARGET}

To calculate Z_{TARGET} for a power rail, you must know the following information:

- Maximum transient current requirements for all devices in the system that are powered by the power rail under consideration. You can get this information the from manufacturers of the respective devices.

  Note: You can download the Early Power Estimator (EPE) tool to estimate power consumption for all its FPGAs and CPLDs.
- Maximum allowable AC ripple on the power rail as a percentage of the supply voltage. You can get this information from the power supply tolerance specifications of the devices being supplied by the power rail under consideration.
- Z_{TARGET}= [VoltageRail (%Ripple/100)/MaxTransientCurrent]
- Z_{TARGET}= [(1.1)(0.05)/1.5]=36.7mΩ

If this information is available, then Z_{TARGET} can be calculated as:

- Z_{TARGET}= [VoltageRail (%Ripple/100)/MaxTransientCurrent]

For example, to reliably decouple a 1.1-volt power rail that allows 5 percent of AC ripple and is expected to supply a maximum transient current of 1.5 A, the target impedance is:

- Z_{TARGET}= [(1.1)(0.05)/1.5]=36.7mΩ

1.1.3. Determining the f_{TARGET}

The highest frequency of interest is the point in frequency after which adding a reasonable number of decoupling capacitors does not bring down the power rail impedance (ZEFF) below the target impedance (ZTARGET) due to the dominance of the parasitic planar spreading inductance and package mounting inductances. Typically, this f_{TARGET} ranges from 50/60 MHz to 150/200 MHz. Beyond these frequencies, it is expected that power integrity is maintained and performed by the on-package and on-die capacitors of the selected target devices.

1.1.4. Select Decoupling CAPS to Meet Z_{TARGET}

To maintain power integrity throughout the entire frequency range of interest, the Power Distribution System relies on the voltage regulator module (VRM), the on-board discrete decoupling capacitors, and inter-plane capacitance (capacitance from the power-ground sandwich in the board stack-up). For the above example, the designer must select these appropriately so that the effective impedance remains below 36.7 Ω within the entire frequency range of interest.

For more details on applying the FDTIM, refer to the PDN Decoupling Calculator Tool and its corresponding User Guide.

1.2. Voltage Regulator Selection

- Voltage Regulator Selection for FPGAs White Paper (PDF)

1.3. PDN Design Tool

Power Delivery Network (PDN) tool does not support Intel® Agilex™ devices (Intel Agilex devices PDN design guideline is based on time domain simulation and worst step load at package level; however, PDN tool design guideline is based on frequency simulation and target impedance).

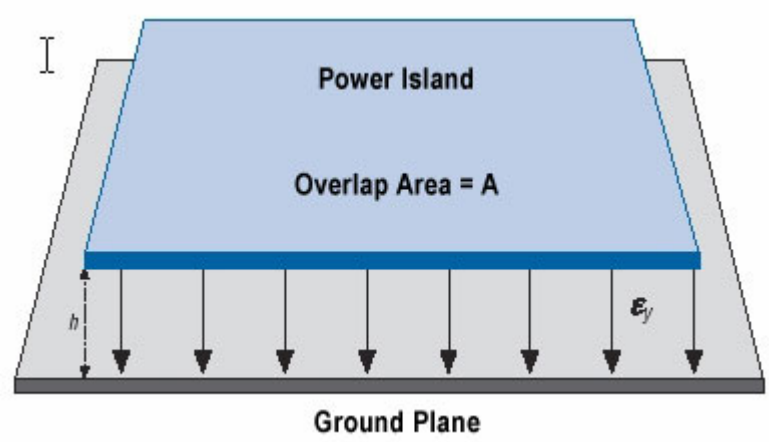
- AN 910: Intel Agilex Power Distribution Network Design Guidelines
- Power Delivery Network (PDN) Tool User Guide (PDF)
  - Power Delivery Network (PDN) Tool (ZIP)
- Device-Specific Power Delivery Network (PDN) Tool User Guide for Stratix® V, Arria® V, Arria II GZ, Cyclone® V, and Cyclone IV Device Families (PDF)
  - Power Delivery Network (PDN) Tool for Arria V, Stratix V, Cyclone IV, and Arria II GZ Devices (ZIP)
- Device-Specific Power Delivery Network (PDN) Tool User Guide (PDF)
  - Power Delivery Network (PDN) Tool for Arria II GX Devices
  - Power Deliver Network (PDN) Tool for Stratix IV Devices
  - Power Deliver Network (PDN) Tool for Stratix III Devices
- Device-Specific Power Delivery Network (PDN) Tool 2.0 User Guide (PDF)
  - Power Delivery Network (PDN) Tool 2.0 for Intel Stratix 10, Intel Arria 10, and Intel Cyclone 10 devices
  - Power Delivery Network (PDN) Tool 2.0 for Intel MAX® 10 Devices

    Note: For Intel Arria 10 devices, do not use the previous version of the Intel Arria 10 combined in Intel MAX 10 PDN Tool. The latest version of Intel Arria 10 and Intel Stratix 10 is already available through the link above.
  - Power Delivery Network (PDN) Tool 2.0 for Stratix V, Arria V, Cyclone V, Cyclone IV, and Arria II GZ Devices (ZIP)

1.4. Plane Capacitance

For high frequencies, decoupling using discrete capacitors is less effective. Use power plane capacitance for decoupling noise at these frequencies. You can understand the concept of plane capacitance by studying the classic parallel plate capacitor, as shown in Figure 1 on page 8.

Figure 1. Parallel Plane Capacitance



An electric field is created when there is a power plane next to a ground plane. The upper area in Figure 1 on page 8 shows the power island or plane, the lower area shows the ground plane, and the arrows represent the electric field lines. This electric field gives rise to a capacitance, the magnitude of which is shown by the following equation:

$C=(\varepsilon o \varepsilon r A)/h$

Where

- εo = permittivity of free space
- εr = relative permittivity of the dielectric used
- A = area of overlap
- h = separation of the planes.

If there are ground planes on both sides of the power island, then the capacitance needs to be calculated for each side and added to determine the total capacitance.

Plane capacitance is the primary way of decoupling at high frequencies, so it must be an integral part of any high-speed design. At high frequencies, the discrete capacitors are not very effective.

As an example, consider the following.

Example: Determine the parallel plate capacitance for 1 square inch of area overlap in an FR-4 dielectric (εr = 4.5) and a separation of 4 mils.

Solution:

Applying these numbers to the capacitance equation above yields C = 253 pF. Therefore, there is about 253 pF per square inch of area overlap on a typical FR-4 board with 4 mils of separation. The value scales inverse linearly with separation and linearly with area.

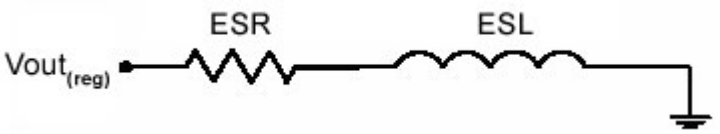
1.5. Minimization Parasitic Inductances

The goal of a Power Distribution System (PDS) is to provide and maintain a target constant voltage at the power and ground pads of each device. To effectively accomplish this, the PDS relies on the Voltage Regulator Module (VRM), the bulk and decoupling capacitors (Decaps) and the power and ground plane sandwich (plane capacitance). How effectively each of these components perform their job in helping to maintain a constant voltage under varying transient loads depends highly on their associated parasitic inductances.

1.5.1. VRM

As a first-order approximation, the VRM can simply be modeled as a series connected resistor and inductor, as shown in Figure 2 on page 9.

Figure 2. Series Impedance Model of a VRM



At low frequencies in the range of tens of KHz, the VRM, being primarily resistive, provides very low impedance and thus it is capable of providing the instantaneous current requirements at these lower frequencies. However, beyond a few tens of KHz, the VRM impedance being primarily inductive makes it incapable of providing the transient current requirements. You can get the ESR and ESL values of the VRM from the VRM manufacturer and you can choose a low ESR/ESL regulator for best transient performance.

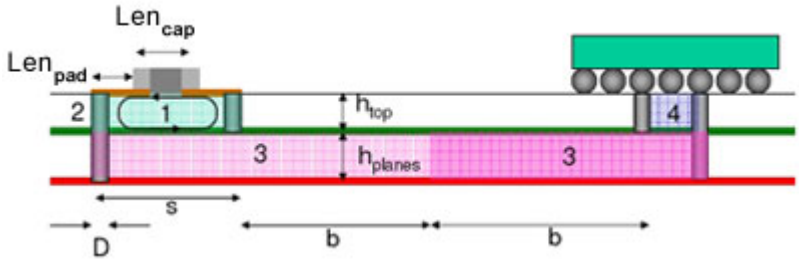
1.5.2. Decaps

The on-board discrete decoupling capacitors must provide the required low impedance from a few tens of KHz to about a couple of hundred MHz (maximum) depending upon the capacitor ESR and ESL as well as board mounting and spreading inductance parasitics. Even choosing decoupling capacitors with very low ESR and ESL specifications is not enough as the contribution of the parasitic mounting and spreading inductances can limit the effectiveness of these capacitors. Hence, to design an effective PDS, care must be taken to minimize the various parasitic inductances associated with the design of the board.

1.5.3. Mounting Inductance

Mounting Inductance is the additional series inductance contribution associated with the mounting of the capacitor on the PCB. This parasitic inductance adds to the published ESL value that is provided by the cap vendor. Mounting inductance can be minimized by choosing smaller capacitor packages and performing proper layout of the capacitor on the PCB. Figure 3 on page 10 shows the cross-section of a mounted decoupling capacitor in relation to the PCB planes and BGA device.

Figure 3. Decoupling Cap Mounting



To estimate the mounting inductance, use the following equation:

- Lmnt = Ltrace + Lvia

Where,

- Ltrace = 128*[(2xLenpad)+Lencap]*(htop/w) pH

And,

- Lvia = 10*htop*ln(2s/D) ph

Where,

- Lenpad = Length of the capacitor pads plus trace length from pad to the via (mils)
- Lencap = Length of the capacitor body (mils)
- w = Width of the trace between the capacitor pad and via (mils)
- htop = Distance between the top layer and the nearest power/ground plane (mils)
- s = Distance between the capacitor's power via center and ground via center (mils)
- D = Outer diameter of the via (mils)
- hplanes = Distance between the power and ground plane (mils)
- b = Half the distance between the capacitor and the package power/ground vias (mils)

Generally, to minimize the mounting inductance, keep the capacitors power and ground vias as close as possible to its respective pad and use wide connecting traces and larger via drill diameters, if possible. Placing power and ground plane pairs closer to the surface where the capacitor is mounted reduces the via inductance contribution. Additionally, placing the vias on the same sides of the capacitor (via-on-side configuration) as opposed to the opposite ends of the capacitor (via-on-end configuration) reduces the current loop area, minimizing the amount of flux lines that penetrate the loop and thus reduces the inductance. Figure 4 on page 11 illustrates the various capacitor layout topologies while Table 1 on page 11 compares mounting inductance of those various cap layout styles for different size capacitors.

Figure 4. Various Capacitor Layout Topologies

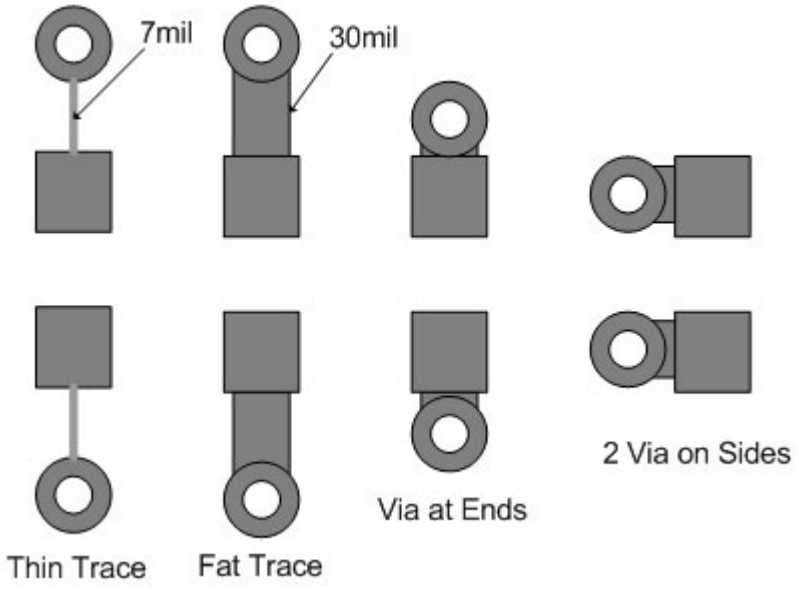


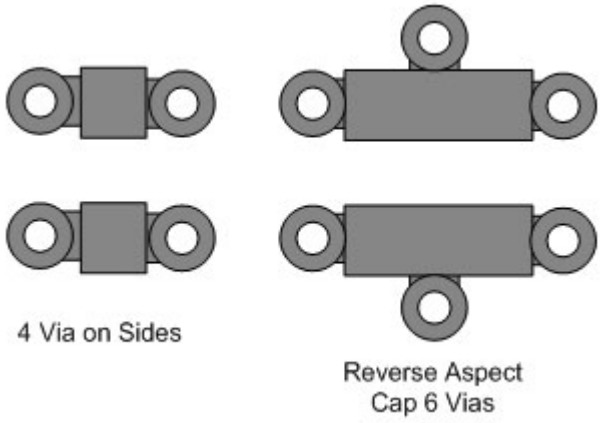
Table 1. Mounting Inductance Comparison

Note: All values are in nano-Henries and drill sizes are in inches.

Via Length (height above plane, inches)	0603 footprint	0603 footprint	0603 footprint	0603 footprint	0402 footprint	0402 footprint
0.020 drill	**Thin trace from pad to via**	**Short thick trace**	**Via on the end of pads**	**Via on the side of pads**	**Via on the end of pads**	**Via on one side of pads**
0.004	1.57	1.04	0.82	0.52	0.8	0.5
0.006	1.96	1.35	1.05	0.65	1	0.63
0.01	2.51	1.87	1.4	0.88	1.34	0.86
0.02	3.45	2.87	2.13	1.39	1.99	1.36
0.010 drill						
0.004	1.61	1.08	0.86	0.56	0.84	0.54
0.006	1.99	1.39	1.08	0.69	1.04	0.67
0.01	2.55	1.92	1.45	0.92	1.38	0.9
0.02	3.49	2.91	2.17	1.44	2.03	1.4

As shown in Table 1 on page 11, the optimum mounting inductance of 0.50 nH is obtained when the smaller 0402 size capacitor is mounted using the via-on-side scheme with wide connecting traces to the larger 20 mil via drill diameter. Figure 5 on page 12 shows other recommended via placement configurations that further reduce mounting inductance. However, these styles require additional vias and hampers routability.

Figure 5. Via placement Styles that Yield Lower Mounting Inductance



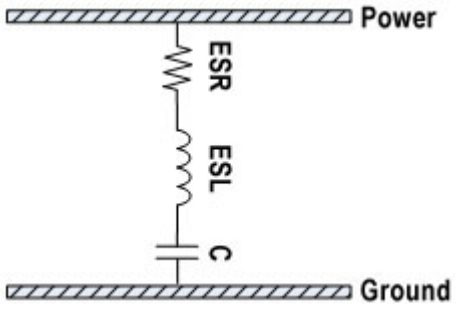
1.5.4. Spreading Inductance

Spreading Inductance is the inductance formed by the loop area between the power-ground plane pair and the distance from the decoupling cap to the power-ground balls of the target BGA device. As a result, this inductance is directly related to the inductance of the inter-plane capacitance formed by the power-ground sandwich. This inductance is explained in detail in the inter-plane capacitance section below.

1.5.5. Inter-Plane Capacitance

As a first-order analysis, the PCB power-ground plane pair can simply be modeled as a series connected resistor, inductor, and, capacitor as shown in Figure 6 on page 12. Note that this simple model ignores the frequency dependent effects such as skin effect and dielectric absorption.

Figure 6. Simplified Impedance Model of the Power-Ground Plane Sandwich



The first-order equations for the ESL in Figure 6 on page 12 is shown below:

- ESL = (μ0•h•l)/w

Where,

- μ0 = magnetic permeability of free space (32 pH/mil)
- h = distance between the power and ground planes in mils
- l = length of the power plane in inches
- w = width of the power plane in inches

You can interpret the ESL of the power-ground plane sandwich as the spreading inductance that the decoupling capacitor sees as it is supplying current to the BGA device. Therefore, from the ESL equation above, the spreading inductance can be reduced by placing the decoupling capacitors as close as possible to the target BGA device (minimizing the distance l from the cap to the BGA device). Additionally, using a thin dielectric material (minimizing h) and wide plane pairs (maximize w) for the power-ground plane sandwich helps reduce the effective spreading inductance seen by the decoupling capacitor.

1.5.6. Conclusion

To successfully design an effective PDS requires an understanding of the various parasitic inductances that can limit the performance of the PDS. This document explains three parasitic inductances that you must be concerned with in designing the PDS. VRM parasitic inductance, decoupling capacitor mounting inductance, and power plane spreading inductance are examined and techniques to minimize them are presented. For additional information and tools, download the PDN Decoupling Calculator tool and user guide

1.6. Additional Resources

- *High-Speed Board Design Advisor: Power Distribution Network* (PDF)
- *Stratix III FPGA Signal Integrity white paper* (PDN section) (PDF)
- Power Distribution Network Design for Stratix III and Stratix IV FPGAs (OPDN1100) *0.5 Hours Online Course*

2. Gigahertz Channel Design Considerations

2.1. Material Selection and Loss

- *AN 528: PCB Dielectric Material Selection and Fiber Weave Effect on High-Speed Channel Routing* (PDF)

2.2. Routing Guidelines for Gigabit Channel Designs

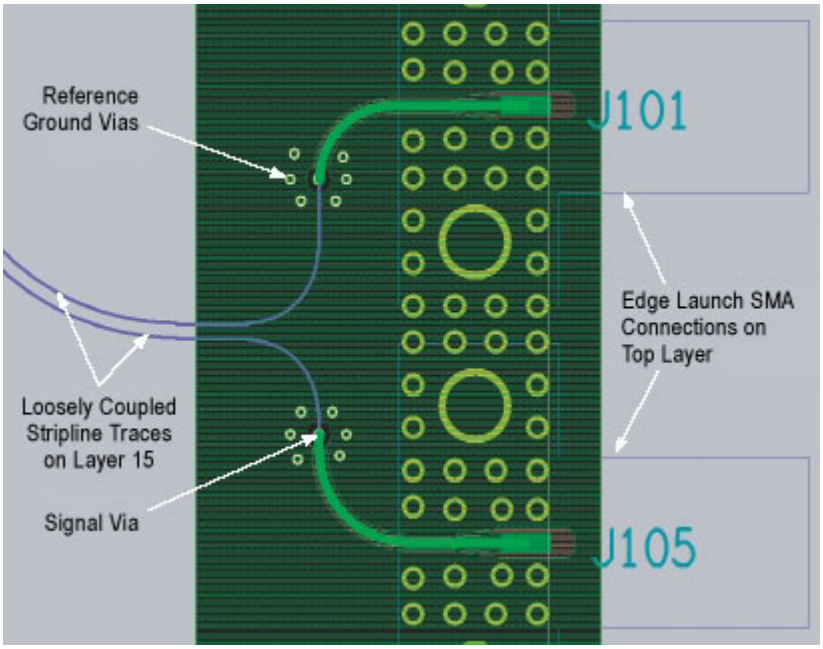
2.2.1. Via Optimization Techniques

2.2.1.1. General Routing Guidelines

This section provides a summary of routing guidelines for laying out high-speed differential pair traces.

Use tight coupling for differential traces as much as possible. If it is not possible to maintain tight coupling throughout the trace length, use loose coupling for the entire trace. Figure 7 on page 15 shows 3.125-Gbps differential transceiver traces, which are loosely coupled on layer 15 and the top layer of the Stratix GX development board. Tight coupling is not possible on the top layer because of the minimum separation required (because of the mechanical constraints of the SMA connectors). If you used tight coupling on layer 15, the signal needs to transition from tight to loose coupling, which introduces impedance discontinuities.

Figure 7. Loose Coupling and Ground Reference Example



- Microstrip and stripline transmission lines are both fine for routing. Stripline has the added benefit of shielding the signals from other noise sources. The constraints during layout design often dictate on which layer to do the routing.
- Use rounded corners while routing. Do not use 90° bends, which introduce impedance discontinuities. A 45° bend provides a compromise, but rounded corners offer the best performance.
- Both broadside and edge coupling are fine. Generally, broadside coupling makes it easy to route out of the BGA, but requires more layers. Edge coupling makes it harder to route out of the BGA, but requires fewer layers. Both provide acceptable signal integrity performance.
- Do not allow high-speed signals to cross over plane splits. A crossover causes signals to see a longer return path, which increases trace inductance. The increased inductance changes the impedance of the line, causing signal integrity problems.
- Remove all unused via pads during the manufacturing process. Some fabrication houses refer to unused via pads as "nonfunctional via pads." Unused via pads add additional capacitance on the signal path. Figure 8 on page 16 shows examples of unused via pads.

Figure 8. Unused Via Pads

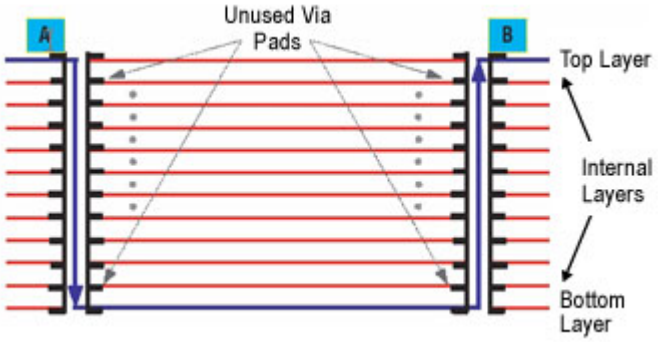
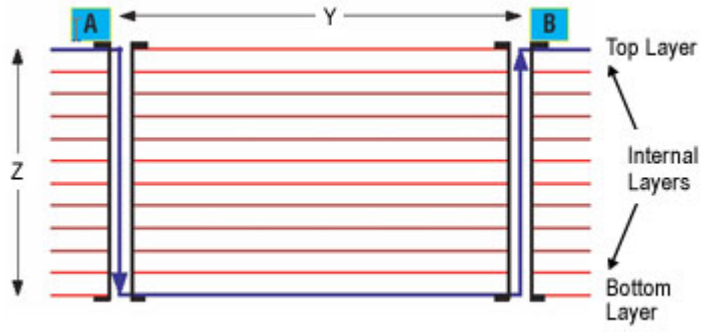
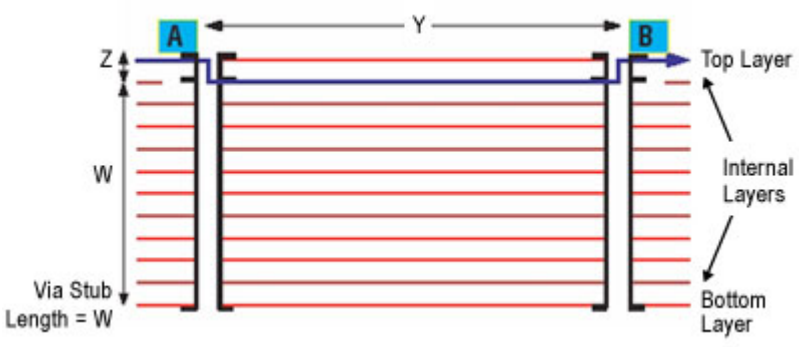


Figure 9. Optimal Routing Path with Through-Hole Vias



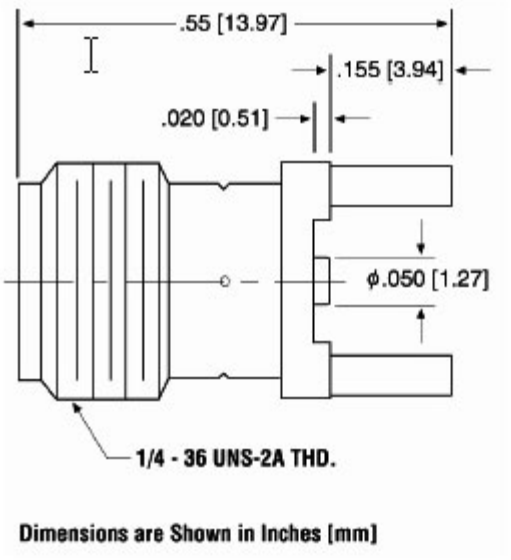
If the signal must be routed through a via, route it so that the via stub length is minimized. The option shown in Figure 9 on page 16 minimizes the via stub length and is preferred over the option shown in Figure 10 on page 16. This technique is used in the Stratix GX development board. The high-speed traces were taken to layer 16 from the top layer through the vias to use as much of the via length as possible and to minimize the stubs.

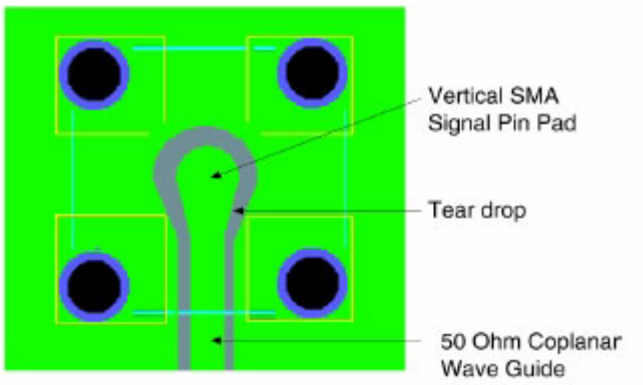
Figure 10. Sub-Optimal Routing Path with Through-Hole Vias



- For dual stripline traces (Power 1 – Signal 1 – Signal 2 - Power 2 configuration), ensure that the signals on the Signal 1 layer are orthogonal to the signals on the Signal 2 layer if they cross each other. If they do not cross each other, ensure that they are at least 4W away, where W is the width of the traces.
- Use "tear-dropping" to reduce impedance discontinuity when going from a wide pin and trace to a narrow pin or trace. For example, when you interface an SMA connector to a trace, use tear-dropping. Figure 11 on page 17 shows the dimensions of an SMA connector and a screen capture of tear-dropping on the Stratix GX development board. The SMA connector has a 50-mil diameter center pin, but the board traces might be 5-mils wide. Tear-dropping helps minimize the impedance discontinuity.

Figure 11. SMA Tear-dropping





Related Information

AN 529: Via Optimization Techniques for High-Speed Channel Designs (PDF)

2.3. Minimizing Adverse Effects of Material Glass Fiber Weaves

- *AN 528: PCB Dielectric Material Selection and Fiber Weave Effect on High-Speed Channel Routing* (PDF)

2.4. Plane Cut-outs Below Surface Mount Pads

- *AN 530: Optimizing Impedance Discontinuity Caused by Surface Mount Pads for High-Speed Channel Designs* (PDF)

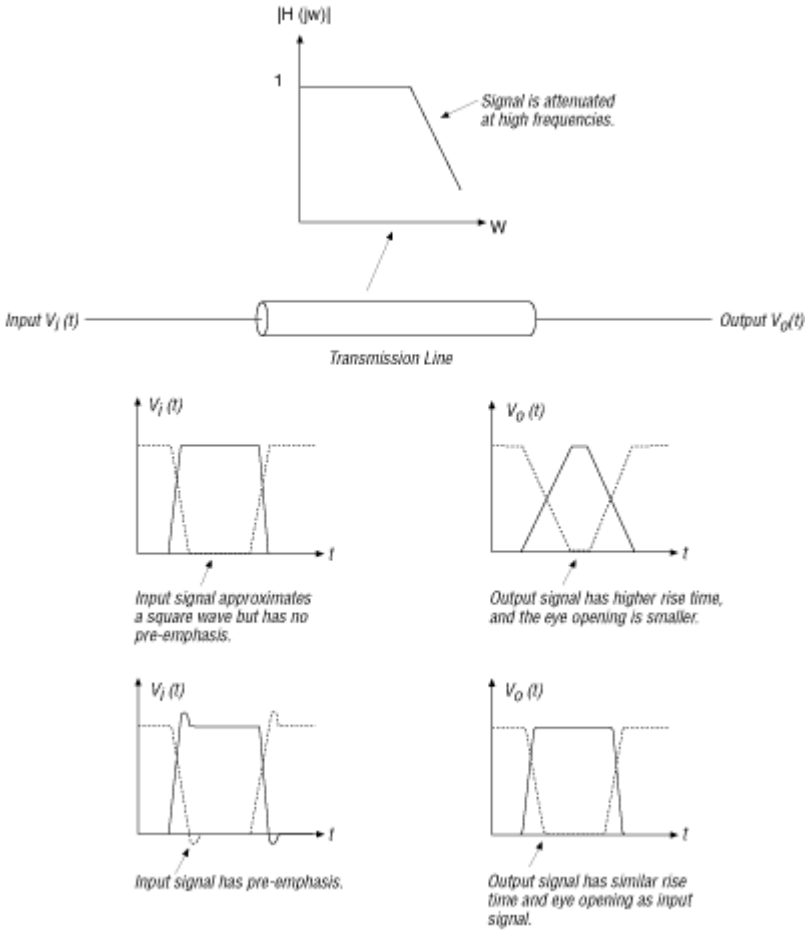
2.5. Transmitter Pre-Emphasis and Receiver Equalization

Typical transmission media like copper trace and coaxial cable have low pass characteristics, so they attenuate higher frequencies more than lower frequencies. A typical digital signal that approximates a square wave contains high frequencies near the switching region and low frequencies in the constant region. When this signal travels through low pass media, its higher frequencies are attenuated more than the lower frequencies, resulting in increased signal rise times. Consequently, the eye opening narrows and the probability of error increases.

The high-frequency content of a signal is also degraded by what is called "skin effect". The cause of skin effect is the high-frequency current that flows primarily on the surface (skin) of a conductor. The changing current distribution causes the resistance to increase as a function of frequency.

You can use pre-emphasis to compensate for the skin effect. By Fourier analysis, a square wave signal contains an infinite number of frequencies. The high frequencies are located in the low-to-high and high-to-low transition regions and the low frequencies are located in the flat (constant) regions. Increasing the signal's amplitude near the transition region emphasizes higher frequencies more than the lower frequencies. When this pre-emphasized signal passes through low pass media, it comes out with minimal distortion, if you apply the correct amount of pre-emphasis. Refer to Figure 12 on page 19 for a graphical illustration of this concept.

Figure 12. Input and Output Signals with and without Pre-Emphasis



Stratix II GX and Stratix IV GX devices provide programmable pre-emphasis to compensate for variable lengths of transmission media. You can set the pre-emphasis to be between 5 percent and 25 percent, depending on the value of the output differential voltage (VOD). Please refer to the Stratix II GX and Stratix IV GX user guides.

Related Information

- PELE Technology
- Differences Between the Stratix GX & Stratix II GX Transceivers
- Stratix II GX Transceiver FPGAs Physical Medium Attachment Layer
- Using Pre-Emphasis and Equalization with Stratix GX white paper (PDF)

2.6. Additional Resources

- *High-Speed Board Design Advisor: High-Speed Channel Design and Layout* technical brief (PDF)
- PELE Technology

3. PCB and Stack-Up Design Considerations

3.1. Material Selection and Loss

- *AN 528: PCB Dielectric Material Selection and Fiber Weave Effect on High-Speed Channel Routing* (PDF)

3.2. Board Thickness and Vias

- *AN 114: Designing With High-Density BGA Packages for Intel Devices* (PDF)

3.3. PCB Recommended Layout Footprint Land Pattern

- *AN 114: Designing With High-Density BGA Packages for Intel Devices* (PDF)

4. Device Pin-Map, Checklists, and Connection Guidelines

4.1. High Speed Board Design Advisor

- Stratix® II GX

4.2. Complete Pin Connection Table by Device

- Pin-Out Files for Intel FPGA Devices

4.3. Pin Connection Guidelines By Device

- Pin Connection Guidelines

4.4. Design for Debug with JTAG Pins

- On-Chip Debugging Resource Center

4.5. Hot Socketing, POR and Power Sequencing Support

- On-Chip Hot-Socketing & Power-Sequencing Support in Intel Devices
- *Altera Hot-Socketing & Power-Sequencing Advantages* white paper (PDF)
- *Hot Socketing and Power-On Reset in Stratix III Devices* (PDF), Chapter 10 of the *Stratix III Handbook*
- *Hot Socketing & Power-On Reset* (PDF), Chapter 4 of the *Stratix II Handbook*
- *Hot Socketing and Power-On Reset in Cyclone III Devices* (PDF), Chapter 11 of the *Cyclone III Handbook*
- *Hot Socketing & Power-On Reset* (PDF), Chapter 4 of the *Cyclone II Handbook*
- *Hot Socketing and Power-On Reset in MAX II Devices* (PDF), Chapter 4 of the *MAX II Handbook*

4.6. Implementing OCT

- *AN 465: Implementing OCT Calibration in Stratix III Devices* (PDF)
- *AN 384: Using Calibrated On-Chip Series Termination in Stratix II Devices* (PDF)

4.7. Unused I/O Pins Guidelines

To allow flexibility in board design, you can specify the state of unused pins as one of the following five states in the Quartus® Prime or Quartus II software: as inputs that are tri-stated, as outputs that drive ground, as outputs that drive an unspecified

signal, as input tri-stated with bus-hold, or as input tri-stated with weak pull-up resistors. To improve signal integrity, set unused pins as outputs that drive ground and tie them directly to the ground plane on the board. Doing so reduces inductance by creating a shorter return path and reduces noise on the neighboring I/O. To reduce power dissipation, set clock pins to drive ground and set other unused I/O pins as inputs that are tri-stated. To make the setting that is appropriate for your design, choose one of the five allowable states for **Reserve all unused pins** on the**Unused Pins** tab in the **Device & Pin Options** dialog box, or apply the **Reserve Pin** assignment to specific pins in the Pin Planner.

When you compile your design, the software generates the pin report file (**.pin**) to specify how you should connect the device pins. Unused I/O pins are marked in the report file according to the unused pins option you set in the software. All I/O pins specified as GND* can either be connected to ground to improve the device's immunity to noise, or left unconnected. Leave all RESERVED I/O pins unconnected on your board because these I/O pins drive out unspecified signals. Tying a RESERVED I/O pin to VCC, ground, or another signal source can create contention that can damage the device output driver. You can connect RESERVED_INPUT I/O pins to a high or low signal on the board, while RESERVED_INPUT_WITH_WEAK_PULLUP and RESERVED_INPUT_WITH_BUS_HOLD pins can be left unconnected.

4.8. Device Breakout Guidelines

- FineLine BGA Gerber Files & Layout Guidelines
- *High-Speed Board Design Advisor: Pinout Definition* technical brief (PDF)

4.9. Additional Resources

- *High-Speed Board Design Advisor: Pinout Definition* technical brief (PDF)

5. General Board Design Considerations/Guidelines

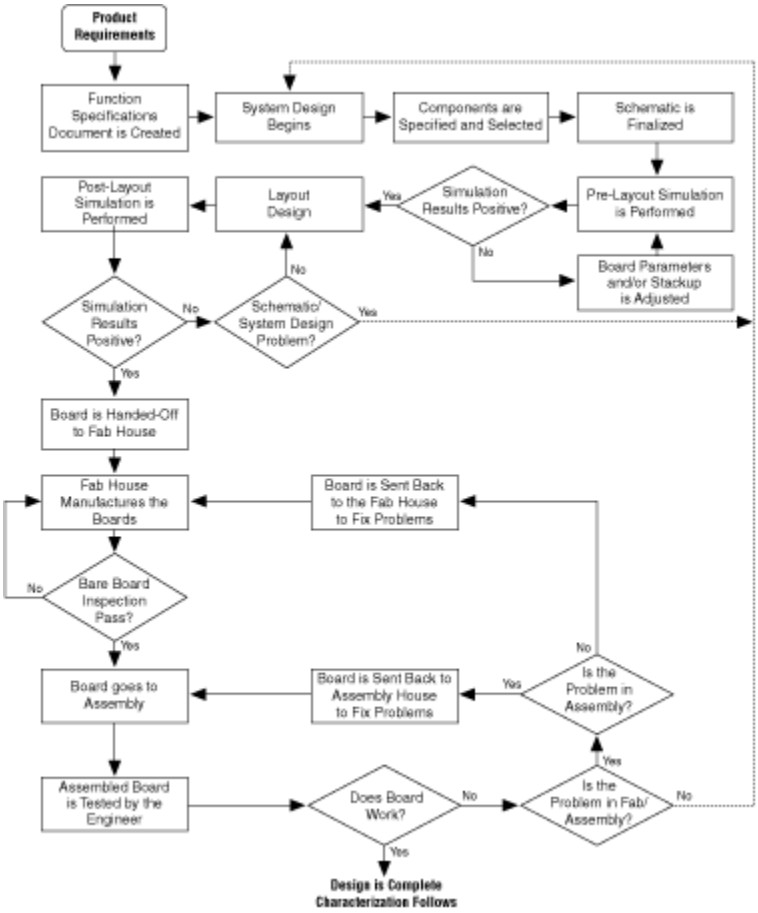
5.1. Board Design Process Overview

A typical high-speed board design begins with a product requirements document, a concept review, and a functional specification. After the specification is finalized, actual system design begins with the selection of key components. At this point, timing analysis for all the interfaces and power analysis — to determine the power requirements — is performed. Based on the power analysis, power supply modules and regulators are chosen, and a decoupling scheme is specified. Based on the timing analysis, a length-matching criterion for the buses is established. Upon completion of these tasks, as well as completion of the system design and selection of remaining components, the schematics are created. These are typically reviewed among engineers who make any necessary changes, after which they are given to the layout designer. A layout guidelines document, which specifies how to place components on the board and how to route the traces, is created and given to the layout designer as well.

A pre-layout simulation is performed to determine the ideal stack-up, trace widths, spacing, and other routing requirements. Any changes to the schematic, based on the simulation results, are incorporated in the schematic and provided to the layout designer. When the layout is complete, a post-layout simulation is performed on the critical sections of the board to ensure that there are no major signal integrity problems.

Based on the results of the post-layout simulation, any changes required are incorporated into the layout, and finally the layout is released to the fabrication house for board manufacturing. Figure 13 on page 24 shows typical board design process.

Figure 13. Simplified View of a Typical High-Speed Board Design Process



5.1.1. Material Selection and Loss

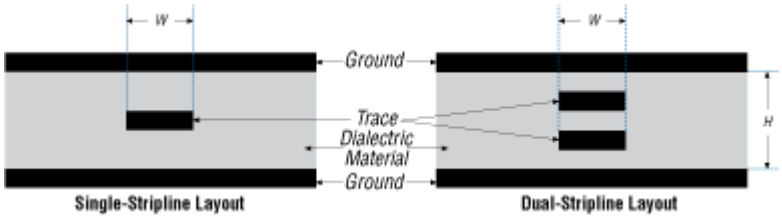
- *AN 528: PCB Dielectric Material Selection and Fiber Weave Effect on High-Speed Channel Routing* (PDF)

5.1.2. Cross Talk Minimization

Crosstalk is the unwanted coupling of signals between parallel traces. Proper routing and layer stack-up through microstrip and stripline layouts can minimize crosstalk.

To reduce crosstalk in dual-stripline layouts, which have two signal layers next to each other (refer to Figure 14 on page 25), route all traces perpendicular, increase the distance between the two signal layers, and minimize the distance between the signal layer and the adjacent reference plane.

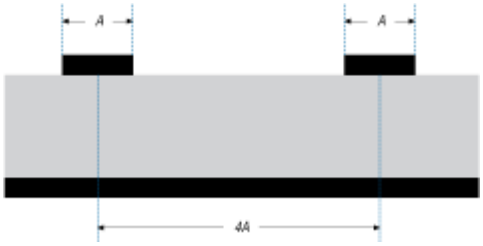
Figure 14. Dual- Stripline Layout



Use the following steps to reduce crosstalk in either microstrip or stripline layouts:

Crosstalk also increases when two or more single-ended traces run parallel and are not spaced far enough apart. The distance between the centers of two adjacent traces should be at least four times the trace width, as shown in Figure 15 on page 25. To improve design performance, lower the distance between the trace and the ground plane to under 10 mils without changing the separation between two traces.

Figure 15. Separating Traces for Crosstalk



- Widen spacing between signal lines as much as routing restrictions allow. Try not to bring traces closer than three times the dielectric height.
- Design the transmission line so that the conductor is as close to the ground plane as possible. This technique couples the transmission line tightly to the ground plane and help decouple it from adjacent signals.
- Use differential routing techniques where possible, especially for critical nets (i.e., match the lengths as well as the gyrations that each trace goes through).
- If there is significant coupling, route single-ended signals on different layers orthogonal to each other.
- Minimize parallel run lengths between single-ended signals. Route with short parallel sections and minimize long, coupled sections between nets.

5.1.3. Power Filtering/Distribution

Clean, evenly distributed power to VCC on all boards and devices can reduce system noise.

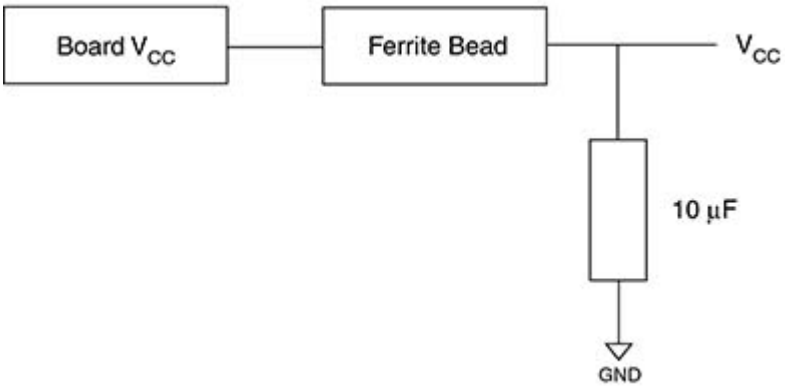
5.1.3.1. Filtering Noise

To diminish the low-frequency (less than 1 kHz) noise caused by the power supply, filter the noise on the power lines at the point where the power connects to the PCB and to each device. You may place a 100-µF electrolytic capacitor adjacent to the location where the power supply lines enter the PCB. If using a voltage regulator,

place the capacitor immediately after the final stage that provides the VCC signal to the device(s). Capacitors not only filter low-frequency noise from the power supply, but also supply extra current when many outputs switch simultaneously in a circuit.

Another way to filter power supply noise is to place a non-resonant surface-mount ferrite bead, big enough to handle the current, in series with power supply. Place a 10-µF to 100µF bypass capacitor next to the ferrite bead (refer to Figure 16 on page 26). Proper terminations, layouts, and filtering in a design eliminate the need for the ferrite bead. In this case, use a 0-Ω resistor instead of the ferrite bead.

Figure 16. Filtering Noise with Ferrite Bead



PCB elements add high-frequency noise to the power plane. To filter high-frequency noise at the device, place decoupling capacitors as close as possible to each VCC and ground pair. Placing the power and ground planes in parallel and separated by dielectric material provides another level of bypass capacitance. These parallel planes reduce the power-related high-frequency noise, since this type of capacitance has no effective series resistance (ESR) and no lead inductance.

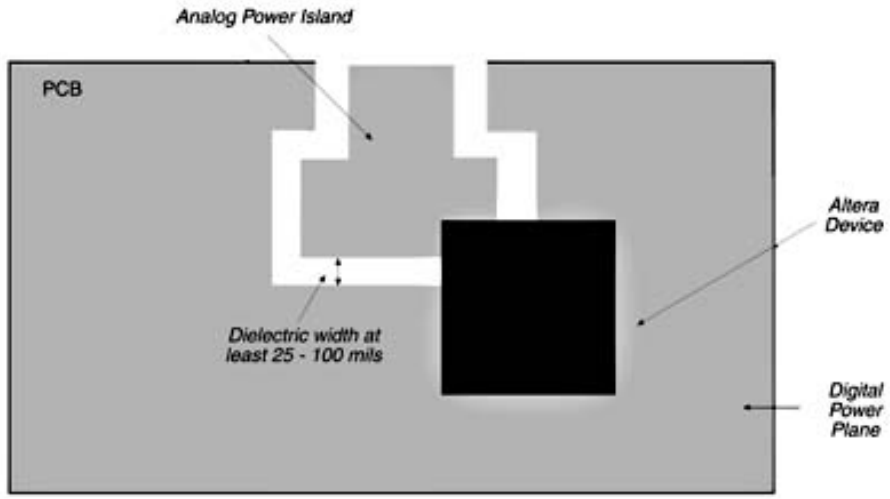
5.1.3.2. Distributing Power

Power distribution also impacts system noise. Either a power bus network or power planes can distribute power throughout the PCB. Usually used on two-layer PCBs, the least expensive way to distribute power is a power bus network, which consists of two or more wide metal traces that carry VCC and ground to the devices. The density of the PCB limits the trace widths, which should be as wide as possible. Power buses have significant DC resistance; the last element on the bus may receive VCC power degraded by as much as 0.5 V.

It is recommended to use power planes to distribute power. Used on multi-layer PCBs, power planes consist of two or more metal layers that carry VCC and ground to the devices. Because the power plane covers the full area of the PCB, its DC resistance is very low. The power plane maintains VCC, distributes it equally to all devices, and provides very high current-sink capability, noise protection, and shielding for the logic signals on the PCB. Sharing the same plane between analog and digital power supplies may be risky due to unwanted interaction of these two circuit types.

For fully digital systems without separate analog power and ground planes on the board, adding two new planes to the board may be prohibitively expensive. Instead, you can create partitioned power islands (split plane). Figure 17 on page 27 shows an example board layout with phase-locked loop (PLL) power islands.

Figure 17. Board Layout for General-Purpose PLL Power Islands



To reduce system noise from power distribution:

- Use separate power planes for the analog power supply for equal power distribution.
- Avoid trace and multiple signal layers when routing the PLL power supply.
- Place a ground plane next to the PLL power supply plane. Place analog and digital components only over their respective ground plane.
- Use ferrite beads to isolate the PLL power supply from the digital power supply.

5.1.4. Unused I/O Pins

To allow flexibility in board design, you can specify the state of unused pins as one of the following five states in the Quartus® prime or Quartus II software: as inputs that are tri-stated, as outputs that drive ground, as outputs that drive an unspecified signal, as input tri-stated with bus-hold, or as input tri-stated with weak pull-up resistors. To improve signal integrity, set unused pins as outputs that drive ground and tie them directly to the ground plane on the board. Doing so reduces inductance by creating a shorter return path and reduces noise on the neighboring I/O. To reduce power dissipation, set clock pins to drive ground and set other unused I/O pins as inputs that are tri-stated. To make the setting that is appropriate for your design, choose one of the five allowable states for **Reserve all unused pins** on the**Unused Pins** tab in the **Device & Pin Options** dialog box, or apply the **Reserve Pin** assignment to specific pins in the Pin Planner.

When you compile your design, the software generates the pin report file (**.pin**) to specify how you should connect the device pins. Unused I/O pins are marked in the report file according to the unused pins option you set in the software. All I/O pins specified as GND* can either be connected to ground to improve the device's

immunity to noise, or left unconnected. Leave all RESERVED I/O pins unconnected on your board because these I/O pins drive out unspecified signals. Tying a RESERVED I/O pin to VCC, ground, or another signal source can create contention that can damage the device output driver. You can connect RESERVED_INPUT I/O pins to a high or low signal on the board, while RESERVED_INPUT_WITH_WEAK_PULLUP and RESERVED_INPUT_WITH_BUS_HOLD pins can be left unconnected.

5.1.5. Signal Trace Routing

Proper routing helps to maintain signal integrity. To route a clean trace, you should perform simulation with good signal integrity (SI) tools. The following section describes the two different types of signal traces available for routing:

- Single-ended trace
- Differential pair trace

5.1.5.1. Single Ended Trace Routing

A single-ended trace connects the source and the load/receiver. Single-ended traces are used in general point-to-point routing, clock routing, low-speed, and non-critical I/O routing. This section discusses different routing schemes for clock signals. You can use the following types of routing to drive multiple devices with the same clock.

Use the following guidelines to improve the clock transmission line's signal integrity:

- Daisy chain routing
  - With stub
  - Without stub
- Star routing
- Serpentine routing
- Keep clock traces as straight as possible. Use arc-shaped traces instead of right-angle bends.
- Do not use multiple signal layers for clock signals.
- Do not use vias in clock transmission lines. Vias can cause impedance change and reflection.
- Place a ground plane next to the outer layer to minimize noise. If you use an inner layer to route the clock trace, sandwich the layer between reference planes.
- Terminate clock signals to minimize reflection.
- Use point-to-point clock traces as much as possible.

Daisy Chain - With Stub

Daisy chain routing is a common practice in designing PCBs. One disadvantage of daisy chain routing is that stubs, or short traces, are usually necessary to connect devices to the main bus (refer to Figure 18 on page 29). If a stub is too long, it induces transmission line reflections and degrade signal quality. Therefore, the stub length should not exceed the following conditions:

TDstub < (T10% to 90%)/3

Where TDstub = Electrical delay of the stubT10% to 90% = Rise or fall time of signal edge

For a 1-ns rise-time edge, the stub length should be less than 0.5 inches. If your design uses multiple devices, all stub lengths should be equal to minimize clock skew. Figure 1 shows stub routing. If possible, you should avoid using stubs in your PCB design. For high-speed designs, even very short stubs can create signal integrity problems.

Figure 18. Daisy Chain Routing with Stubs

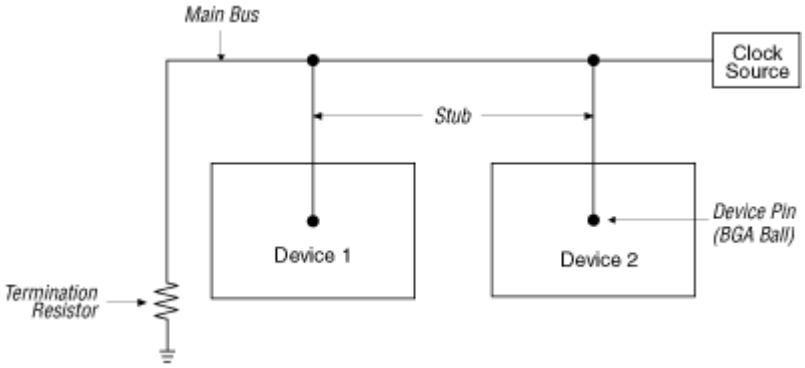


Figure 19 on page 29, Figure 20 on page 29, and Figure 21 on page 30 show the SPICE simulation with different stub length. As the stub length decreases, there is less reflection noise, and the eye opening increases due to less reflection noise.

Figure 19. Stub Length = 0.5 Inch

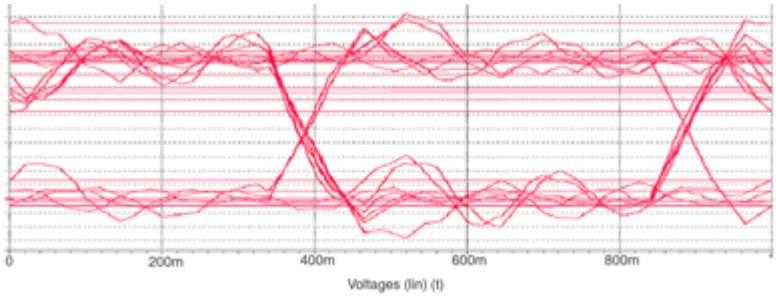


Figure 20. Stub Length = 0.25 Inches

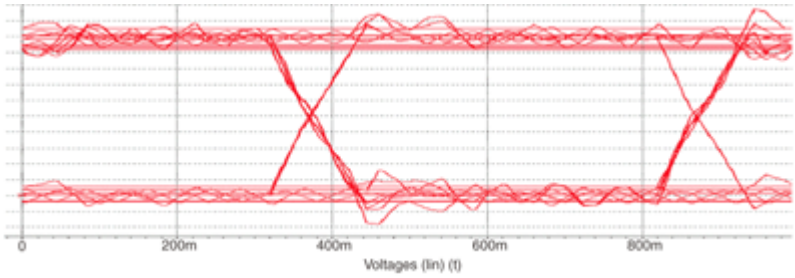
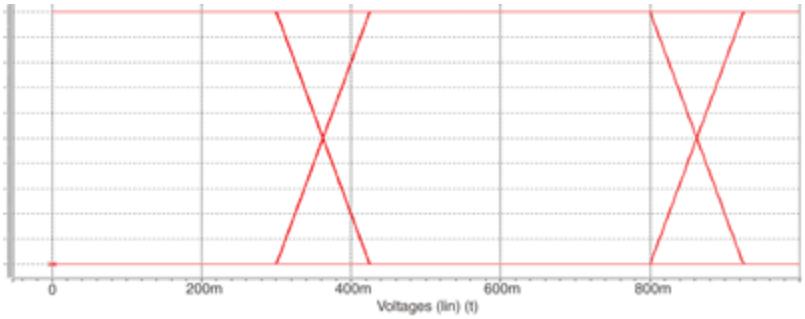


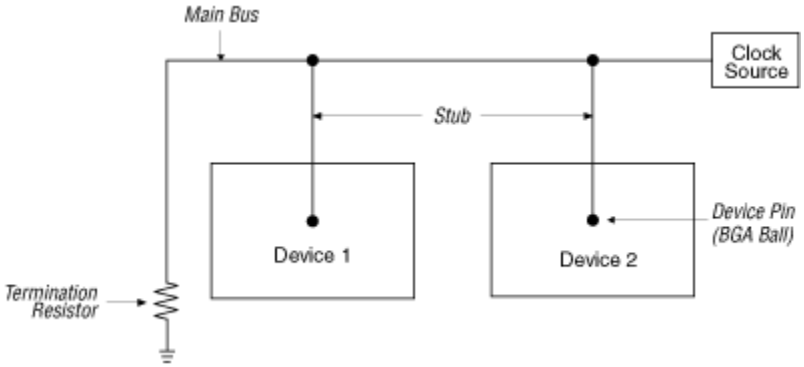
Figure 21. Stub Length = Zero Inches



Daisy Chain - Without Stub

Figure 22 on page 30 shows daisy chain routing with the main bus running through the device pins, eliminating stubs. This layout removes the risk of impedance mismatch between the main bus and the stubs, minimizing signal integrity problems.

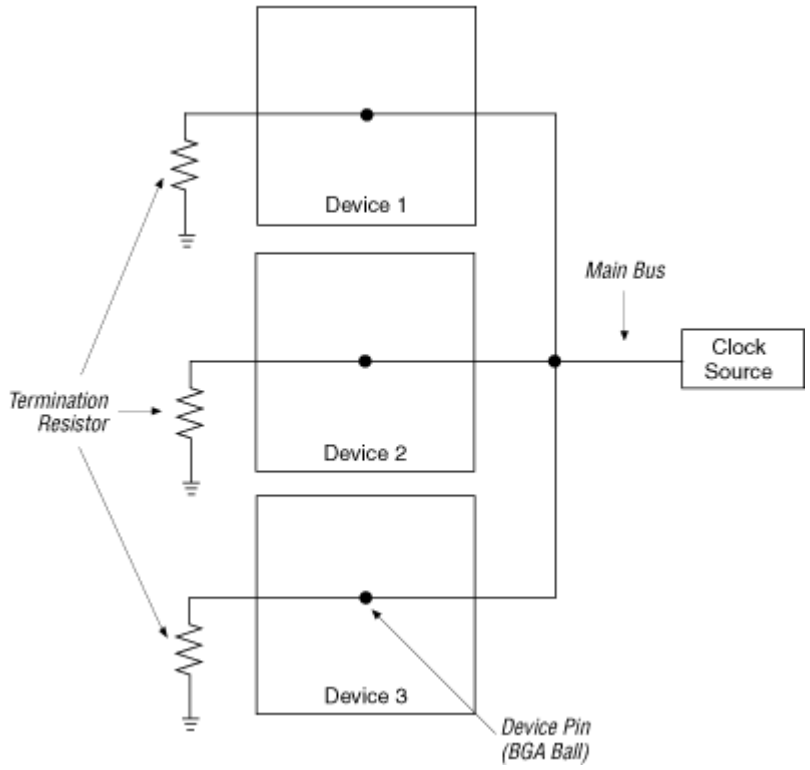
Figure 22. Daisy Chain Routing without Stubs



Star Routing

In star routing, the clock signal travels to all the devices at the same time (refer to Figure 23 on page 31). Therefore, all trace lengths between the clock source and devices must be matched to minimize the clock skew. Each load should be identical to minimize signal integrity problems. In star routing, you must match the impedance of the main bus with the impedance of the long trace that connects to multiple devices.

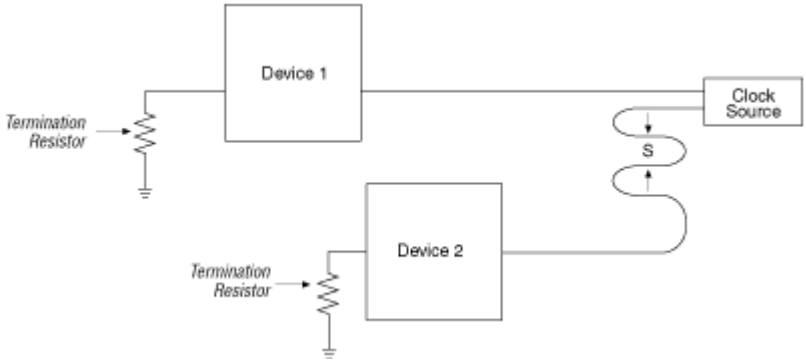
Figure 23. Star Routing



Serpentine Routing

When a design requires equal-length traces between the source and multiple loads, you can bend some traces to match trace lengths (refer to Figure 24 on page 31). Improper trace bends affects signal integrity and propagation delay. To minimize crosstalk, ensure that S ≥ 3 x H, where S is the spacing between the parallel sections and H is the height of the signal trace above the reference ground plane.

Figure 24. Serpentine Routing

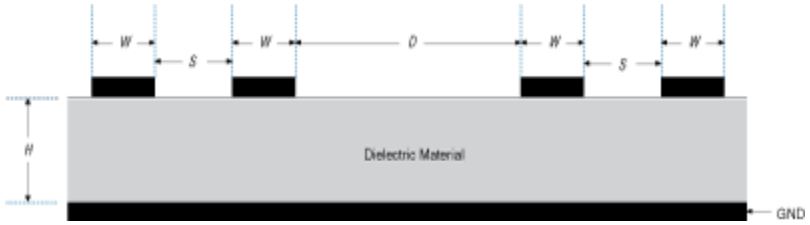


It is recommended to avoid serpentine routing if possible. Instead, use arcs to create equal-length traces.

5.1.5.2. Differential Trace Routing

To maximize signal integrity, proper routing techniques for differential signals are important for high-speed designs. Figure 25 on page 32 shows a differential pair using the microstrip layout.

Figure 25. Differential Trace Routing



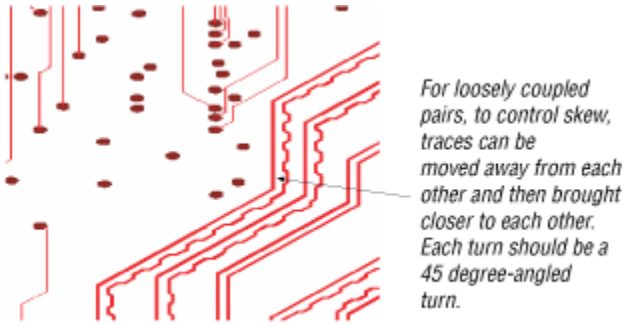
Note: Use the following guidelines when using two differential pairs:

- *D* = Distance between two differential pair signals; *W* = Width of a trace in a differential pair; *S* = Distance between the trace in a differential pair; and *H* = Dielectric height above the group plane.
- Make sure $D > 2S$ to minimize the crosstalk between the two differential pairs.
- To minimize reflection noise, place the differential traces $S = 3H$ as they leave the device.
- Keep the distance between the differential traces (*S*) constant over the entire trace length.
- Keep the length of the two differential traces the same to minimize the skew and phase difference.
- Avoid using multiple vias, because they can cause impedance mismatch and inductance.

5.1.5.3. Skew Minimization

To minimize skew, make sure the two traces of a differential pair are equal lengths. If there is skew between pair traces and if the traces are loosely coupled, the traces can be designed as shown in Figure 26 on page 32. To control the trace length, the traces separate and come back together. Because the traces are loosely coupled, the impedance is just slightly affected.

Figure 26. 45° Turns on the Serpentine Traces



When using serpentine traces, you should have 45° bends (refer to Figure 26 on page 32). Figure 27 on page 33 shows another example using serpentine traces. However, when using the design shown in Figure 27 on page 33, make sure there is no coupling between the adjacent lines. When using serpentine traces for high-speed applications, you should avoid having the traces run parallel to each other at any point. See the example shown in Figure 26 on page 32.

Figure 27. Example of Serpentine Traces

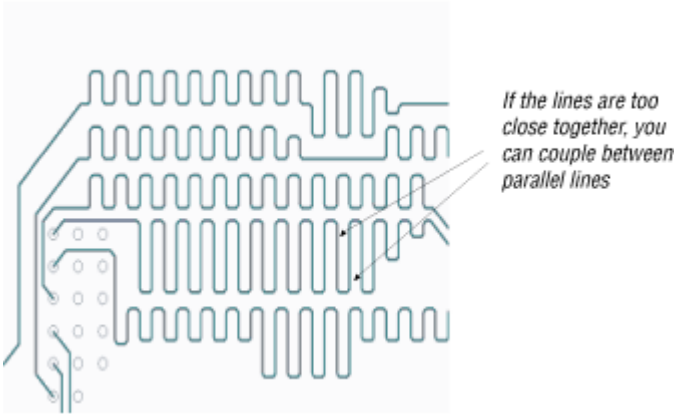
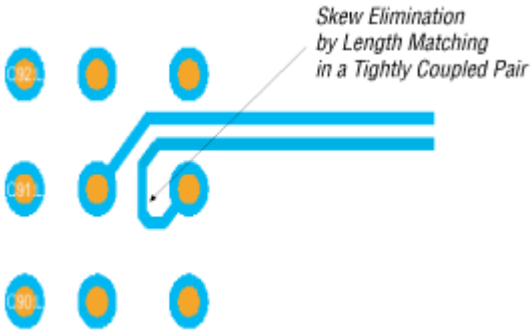


Figure 28 on page 33 shows skew control for tightly coupled pairs. Because the traces are tightly coupled, the impedance changes when the traces are separated and then brought closer together. In a tightly coupled pair, skew-matching is performed at the pin level.

Figure 28. Skew Control Tightly-Coupled Differential Pairs



When designing traces on adjacent signal layers, the traces should not cross each other unless they are almost perpendicular. Parallel traces on adjacent signal layers have coupling between the traces.

5.1.5.4. Termination Schemes

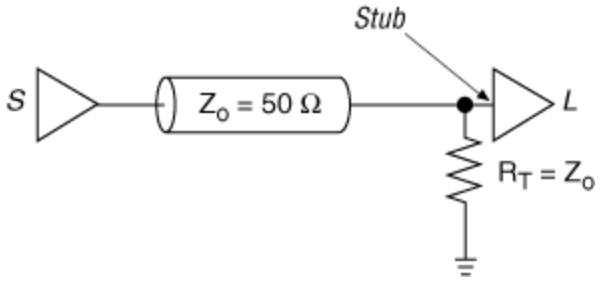
Mismatched impedance causes signals to reflect back and forth along the lines, which causes ringing at the load receiver. The ringing reduces the dynamic range of the receiver and can cause false triggering. To eliminate reflections, the impedance of the source (Z_S) must equal the impedance of the trace (Z_0), as well as the impedance of the load (Z_L). Stratix devices feature support for on-chip implementation of a termination resistor. This section discusses the following signal termination schemes:

- Simple parallel termination
- Thevenin parallel termination
- Active parallel termination
- Series-RC parallel termination
- Series termination
- Differential pair termination
- On-chip termination

5.1.5.4.1. Simple Parallel Termination

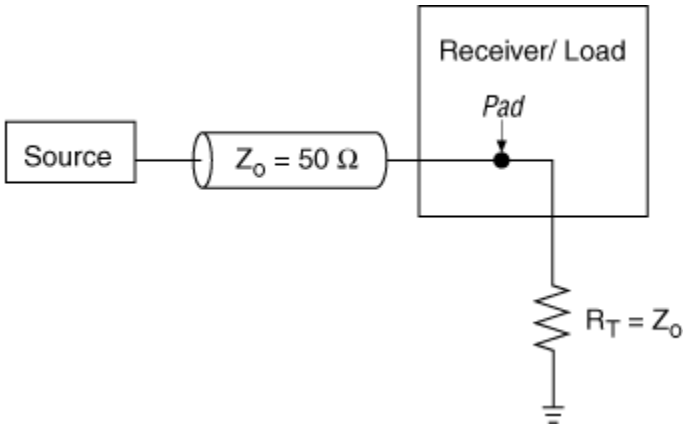
In a simple parallel termination scheme, the terminating resistor (RT) is equal to the line impedance. Place the termination resistor as close to the load as possible to be efficient. Refer to Figure 29 on page 34.

Figure 29. Simple Parallel Termination



The stub length from RT to the receiver pin and pads should be as small as possible. A long stub length causes reflections from the receiver pads, resulting in signal degradation. If your design requires a long termination line between the terminator and receiver, the placement of the resistor becomes important. For long termination line lengths, use fly-by termination (refer to Figure 30 on page 34).

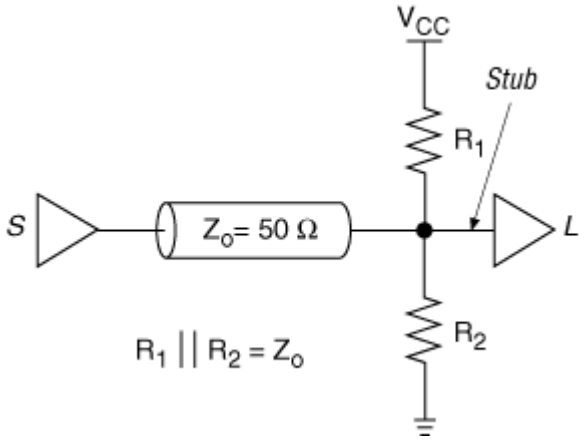
Figure 30. Simple Parallel Fly-By Termination



5.1.5.4.2. Thevenin Parallel Termination

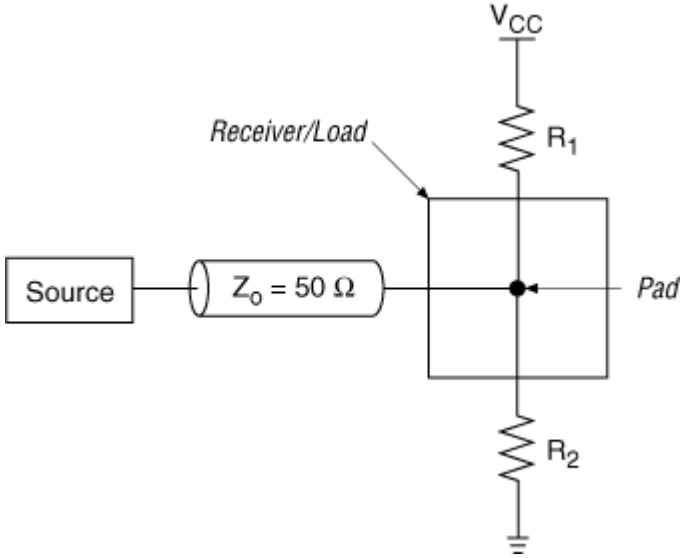
An alternative parallel termination scheme uses a Thevenin voltage divider (refer to Figure 31 on page 35). The terminating resistor is split between R1 and R2, which equals the line impedance when combined. Although this scheme reduces the current drawn from the source device, it adds current drawn from the power supply because the resistors are tied between VCC and GND.

Figure 31. Thevenin Termination



Stub length is dependent on signal rise and fall time and should be kept to a minimum. If your design requires a long termination line between the terminator and receiver, use fly-by termination or Thevenin fly-by termination. Refer to Figure 31 on page 35 and Figure 32 on page 35.

Figure 32. Thevenin Fly-By Termination



5.1.5.4.3. Active Parallel Termination

Figure 33 on page 36 shows an active parallel termination scheme, where the terminating resistor ($R_T = Z_0$) is tied to a bias voltage (VBIAS). In this scheme, the voltage is selected so that the output drivers can draw current from the high- and low-level signals. However, this scheme requires a separate voltage source that can sink and source currents to match the output transfer rates.

Figure 33. Active Parallel Termination

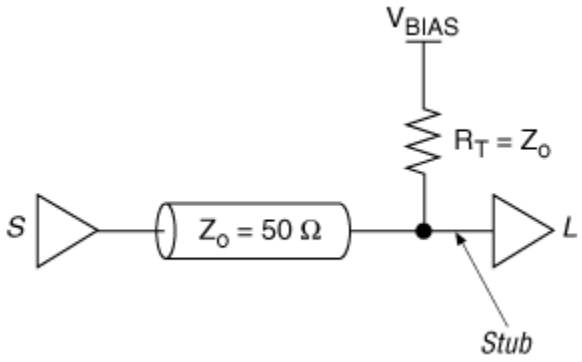
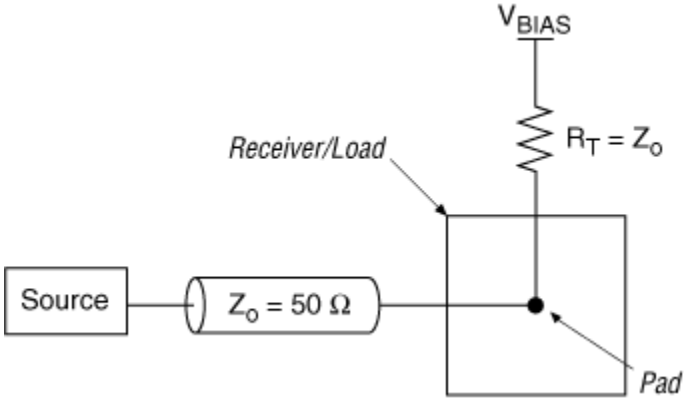


Figure 34 on page 36 shows the active parallel fly-by termination scheme.

Figure 34. Active Parallel Fly-By Termination



5.1.5.4.4. Series-RC Parallel Termination

A series-RC parallel termination scheme uses a resistor and capacitor (i.e., series-RC) network as the terminating impedance. The terminating resistor (RT) is equal to Z0. The capacitor must be large enough to filter the constant flow of DC current. However, if the capacitor is too large, it delays the signal beyond the design threshold.

Capacitors smaller than 100 pF diminish the effectiveness of termination. The capacitor blocks low-frequency signals while passing high-frequency signals. Therefore, the DC loading effect of RT does not have an impact on the driver, as there is no DC path to ground. The series-RC termination scheme requires balanced DC signaling (i.e., the signals spend half the time on and half the time off). AC termination is typically used if there is more than one load. Refer to Figure 35 on page 37.

Figure 35. Series-RC Parallel Termination

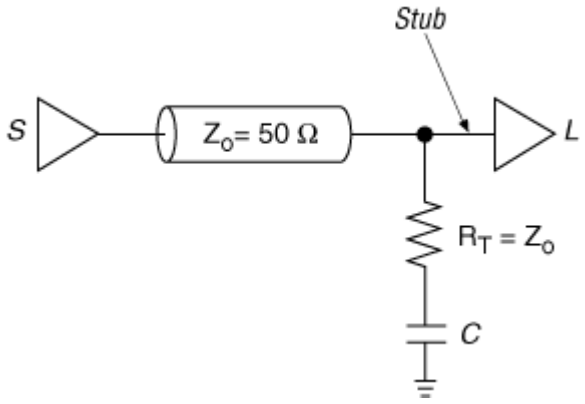
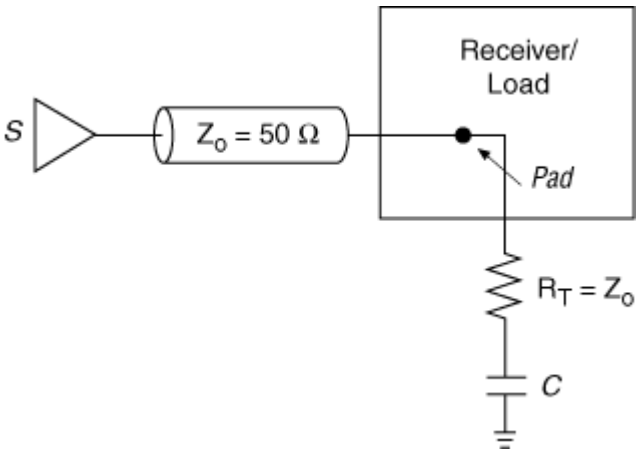


Figure 36 on page 37 shows series-RC parallel fly-by termination.

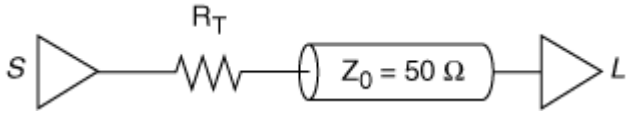
Figure 36. Series-RC Parallel Fly-By Termination



5.1.5.4.5. Series Termination

In a series termination scheme, the resistor matches the impedance at the signal source instead of matching the impedance at each load (refer to Figure 37 on page 37). The sum of RT and the impedance of the output driver should be equal to the Z_0. Because Intel device output impedance is low, you should add a series resistor to match the signal source to the line impedance. The advantage of series termination is that it consumes little power. However, the disadvantage is that the rise time degrades due to the increased RC time constant. Therefore, for high-speed designs, you should perform the pre-layout signal integrity simulation with Intel input/output buffer information specification (IBIS) models before using the series termination scheme.

Figure 37. Series Termination



5.1.5.4.6. Differential Pair Termination

Differential signal I/O standards require a termination resistor between the signals at the receiving device (refer to Figure 38 on page 38). For the LVDS and LVPECL standard, the termination resistor should match the differential load impedance of the bus (i.e., typically 100 Ω). Intel Stratix family of devices, and Mercury™ devices have an on-chip termination option. Using on-chip termination decreases required board space.

Figure 38. Differential Pair (LVDS and LVPECL) Termination

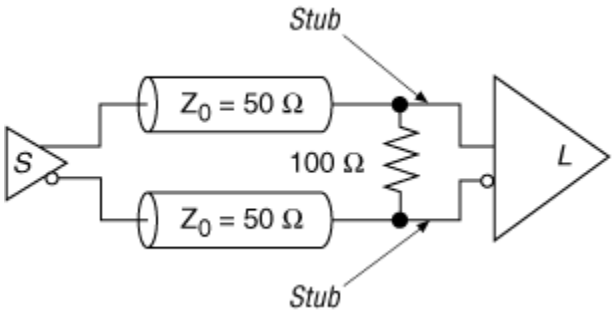
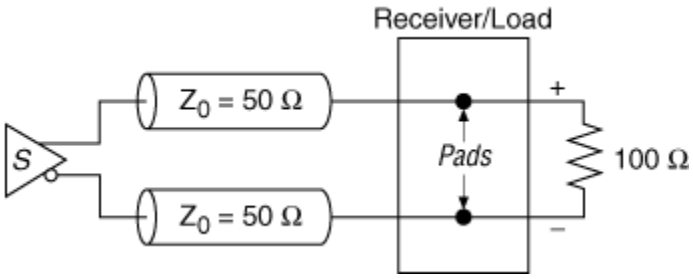


Figure 39 on page 38 shows the differential pair fly-by termination scheme for the LVDS and LVPECL standard.

Figure 39. Pair (LVDS and LVPECL) Fly-By Termination



3.3-V PCML uses two parallel 100-Ω termination resistors at the transmitter and two parallel 50-Ω termination resistors at the receiver (refer to Figure 40 on page 39). The termination voltage (VT) is the same as the VCCIO voltage (3.3 V).

Figure 40. Differential Pair (3.3 V PCML) Termination

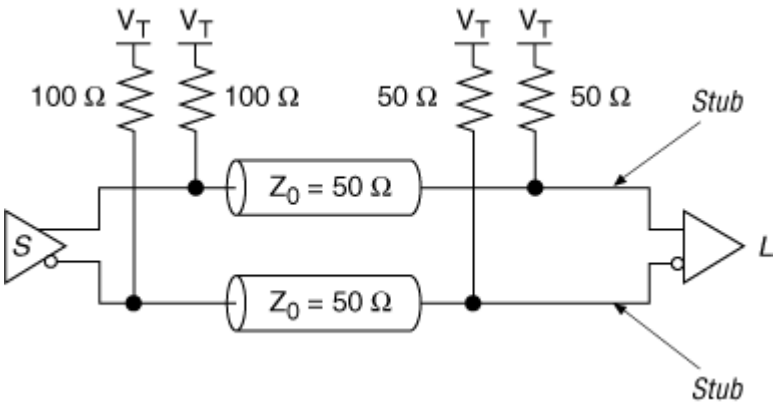
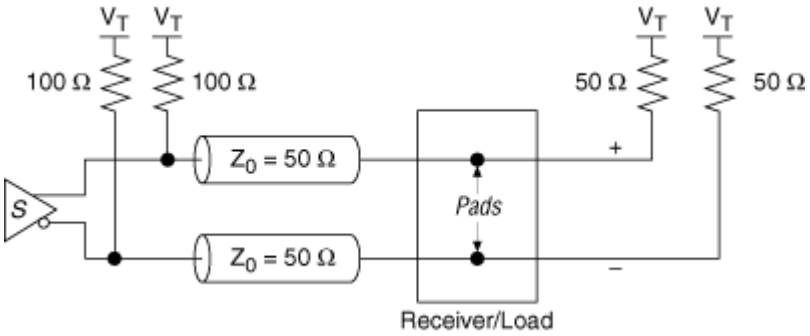


Figure 41 on page 39 shows the differential pair, fly-by termination scheme for 3.3-V PCML.

Figure 41. Differential Pair (3.3 V PCML) Fly-By Termination

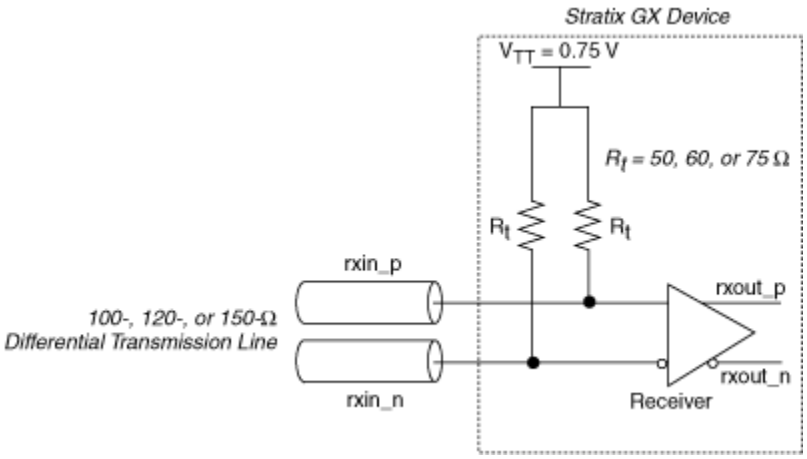


5.1.5.4.7. On-Chip Termination

Stratix family of devices has on-chip resistors designed to support termination for several I/O standards. On-chip resistors simplify the task of board design by freeing board space and offering more freedom in signal routing. Also, using the on-chip resistors reduces stub reflection. Consequently, Stratix series devices provides better load and/or source termination, which leads to better signal integrity.

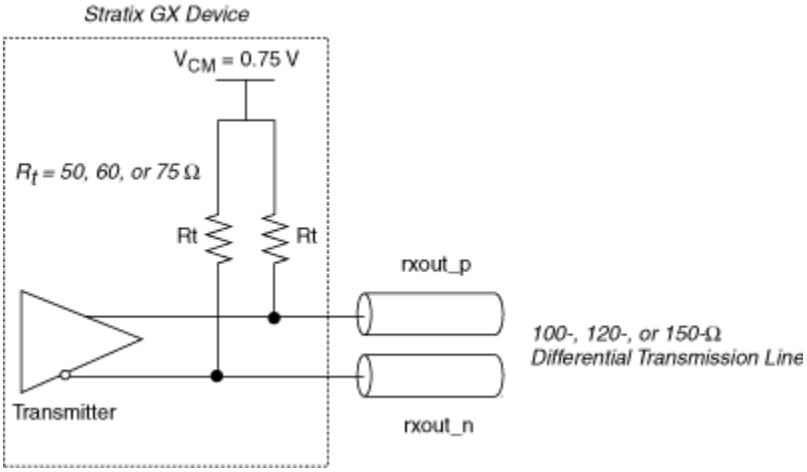
All transceivers in Stratix series devices have programmable, per channel internal termination resistors, which can be programmed to be either 50-, 60-, or 75-Ω single-ended resistors. In differential mode, the resistors generate 100-, 120-, or 150-Ω termination. Figure 42 on page 40 and Figure 43 on page 40 show receiver and transmitter on-chip termination schemes. Because different I/O standards require different termination resistors, programmable internal termination resistors can be helpful. For example, XAUI and Infiniband applications require 100-Ω differential termination, while Gigabit Ethernet and Fibre Channel require 150-Ω differential termination.

Figure 42. Receiver On-Chip Termination



You can bypass the on-chip resistors used in the receiver and use external resistors.

Figure 43. Transmit On-Chip Termination



5.1.5.5. Termination Design Example

When designing a circuit board, one challenge is determining the type of termination to use and where to place it. This section helps you determine the type of termination needed and the best possible location for your custom board design.

If transmission lines associated with data or clock circuitry are not terminated properly, the signal experiences reflections. In this example, the design has the following features:

In this design example, you need to determine if the transmission line should be terminated, and if so, how it should be accomplished.

5.1.5.5.1. Determine the Delay

Use the following equations to determine the delay for a 300-ps rise-time signal traveling through a transmission line embedded in a dielectric with an electrical permittivity of εr.

For stripline configuration:

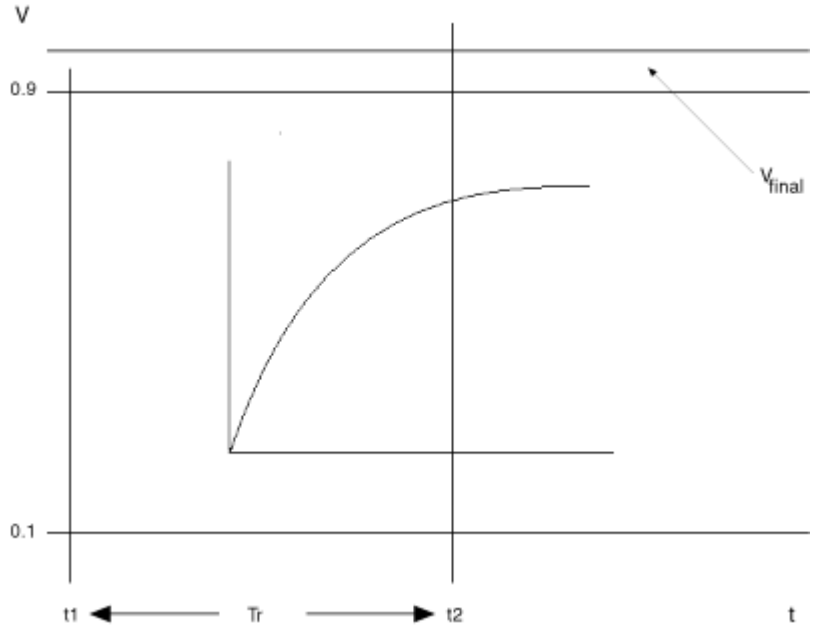
For microstrip configuration:

In FR-4, a transmission line with a stripline configuration induces approximately 180 ps per inch of delay on the signal. Therefore, the velocity of the signal through the transmission line becomes the inverse of the delay, which is $5.5*10^{-9}$ inches per second.

5.1.5.5.2. Determine the Bandwidth

Figure 44 on page 41 shows that the voltage at any instant t is:

Figure 44. Characteristic Voltage Plot for Charging of an AC Circuit



At 10 percent of the curve, you have:

At 90 percent of the curve, you have:

The 10 percent equation divided by the 90 percent equation gives:

Where:

The time constant variable is related to the 3-dB frequency by the equation:

From the previous equation, we can determine the equation for the time constant, r.

Plug the time constant into the voltage equation to get:

Use the following equation to determine the bandwidth for sinusoidal signal

The highest frequency component for a signal with a rise time of Tr have a frequency given by this equation.

The signal previously discussed had a rise time of 300 ps, which means that the highest frequency component present in the signal be:

Using the equation with the bandwidth and speed numbers, you can determine that:

If the transmission line is longer than wavelength/10, then termination is required. In this design example, the transmission line is two inches long, so termination is required.

5.1.5.5.3. Using Series Termination

When using series termination, only near-end termination can be applied. Series termination should only be used with clock signals. With the near-end series termination (Z_0) and the transmission line following, the circuit looks like a voltage divider circuit to the driver, reducing the amplitude V at the driver to V/2 after the series termination. Because there is no termination at the end of the transmission line, when the signal reaches the end, the entire signal reflects being restored to V.

The reflection coefficient is calculated using the following equation:

5.1.5.5.4. Using Parallel Termination

You can place parallel terminations on either ends or only the far end of the transmission line. You should place terminations as close to the source or destination as possible. Any transmission line between the termination and the end of the transmission line appears as a capacitive load to the signal. If you cannot place terminations close to the integrated circuit (IC), place them after the pin (i.e., fly-by configuration).

- A 300-ps rise-time signal
- Two-inch long transmission path between the source and the destination
- *delay* = $85\sqrt{\varepsilon r}$ ps per inch
- *delay* = $85\sqrt{(0.457\varepsilon r+0.67)}$ ps per inch
- $V = V_{FINAL}(1-e^{-t/RC})$
- $0.1\ V_{FINAL} = V_{FINAL}\ (1-e^{-t_1/RC})$
- $0.9 = e^{-t_1/RC}$
- $9 = e^{(t_2-t_1)/RC}$
- $\ln 9 = (t_2-t_1)/RC$
- $2.197 = (t_2-t_1)/RC$
- t_2-t_1 = Rise time of the signal (Tr) and RC = time constant = r
- Frequency = 1/2 пr
- r = RC = 1/2 пf
- 2.197 = 2 пf Tr
- f = 0.35/Tr

- Bandwidth = 0.35/Tr
- Bandwidth = 0.35/300 ps = 1.16 GHz
- Speed = Frequency x Wavelength
- 5.5 giga inches per second = 1.16 GHz x wavelength
- wavelength = 4.74 in.
- wavelength/10 = 0.474 in.
- Reflection coefficient = $(Z_{LOAD} - Z_0)/(Z_{LOAD} + Z_0)$

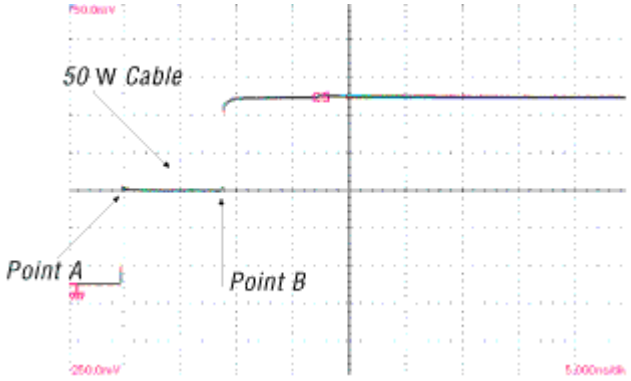
Time Domain Reflectometry

Time domain reflectometry (TDR) is a way to observe discontinuities on a transmission path. The time domain reflectometer sends a pulse through the transmission medium. Reflections occur when the pulse of energy reaches either the end of the transmission path or a discontinuity within the transmission path. From these reflections, the designer can determine the size and location of the discontinuity. This section provides an understanding of TDR.

Figure 45 on page 43 shows a TDR voltage plot for a cable that is not connected to a PCB. The middle line is a 50-Ω cable one meter long. At point A, a pulse starts (Z_0 = 50-Ω) and transmits through the cable, stopping at the end of the transmission line (i.e., Point B). Because the end of the transmission line is open, there is infinite impedance, Z_{LOAD} = ɑ. Therefore, the reflection coefficient at the load is determined with the equation:

The entire signal is reflected. At point B, the amplitude of the signal doubles. Refer to Figure 45 on page 43.

Figure 45. TDR Voltage Plot with Cable Not Connected to PCB



If the same meter-long cable is then connected to a PCB through an SMA connector, the plot changes. Refer to Figure 46 on page 44. Because the SMA connector is more capacitive than inductive in nature, it appears as a capacitive load, shown as a dip in the TDR plot.

Figure 46. TDR Voltage Plot with Cable Connected to PCB

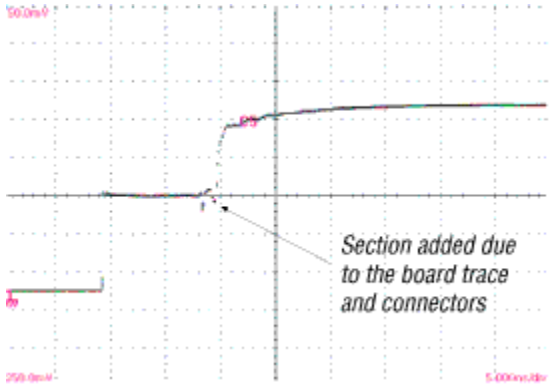
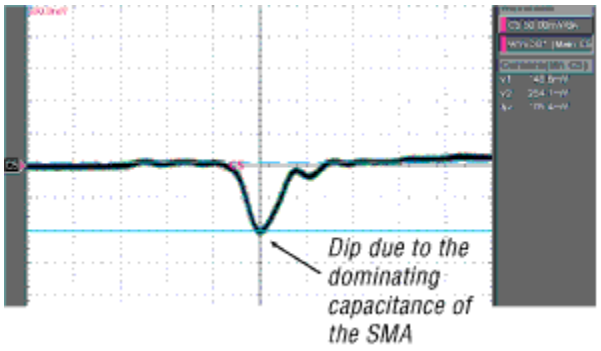


Figure 47 on page 44 shows an expanded curve for the SMA connector. Because the rise time of the pulse sent for TDR analysis is very small (around 20 ps), the TDR voltage plot shows every discontinuity on the transmission path.

The SMA is a capacitive discontinuity on the transmission path, so the signal dips on the voltage plot. The impedance of an ideal transmission line is defined by the following equation:

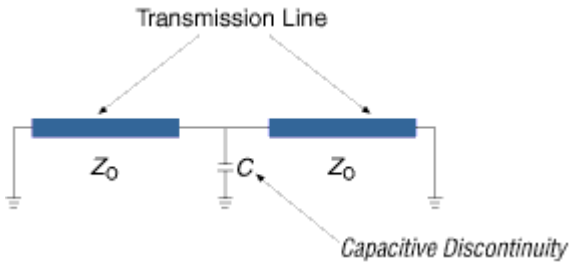
Therefore, when the capacitance increases, the impedance decreases. If the discontinuity is inductive, then the impedance increases, which appears as a bump in the TDR plot. You can calculate the capacitance and inductance from the curves on a TDR plot. If the plot shows a dip, as in Figure 47 on page 44, then calculate the capacitance.

Figure 47. TDR Voltage Plot for the Section Around the SMA Connector on PCB



The equivalent circuit approximation for a dip in the TDR plot is a capacitor to ground, as shown in Figure 48 on page 45.

Figure 48. Equivalent Circuit for a Transmission Line with Capacitive Discontinuity



The RC equation for this type of circuit is:

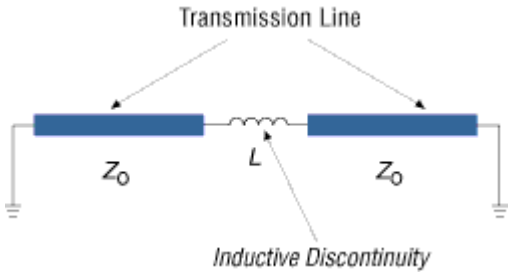
The two transmission lines behave as if they are parallel to each other.

You can determine the change in voltage (ΔV) and rise time (Tr) from the curve. Then, you can enter the values into the equation (i.e., Z_0 = 50 Ω):

Use this equation to determine the RC time constant. You can also use the curve to approximate the RC time constant. Between 0 to 63 percent of the rise is RC. Once the RC is found, you can use it to determine the capacitance (discontinuity, as seen by the signal).

If the discontinuity looks more inductive in nature (i.e., the curve goes up), then the signal experiences a circuit similar to Figure 49 on page 45. The transmission line is split, with an inductive discontinuity in between.

Figure 49. Equivalent Circuit for a Transmission Line with Inductive Discontinuity

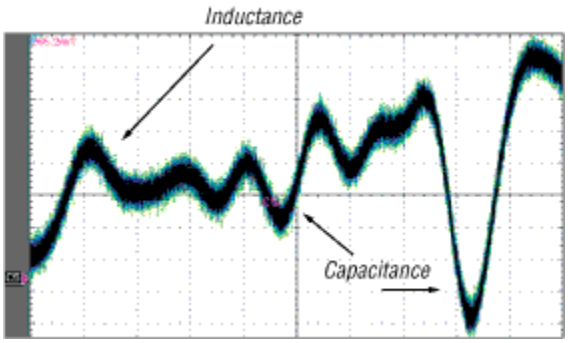


Use the following two equations to find inductive discontinuity (L):

To determine the inductance value, use the equation for (Z_0 = 50 Ω):

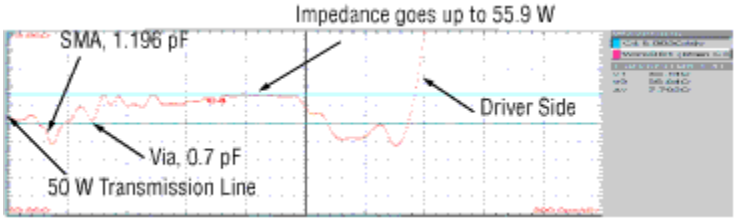
Figure 50 on page 46 shows a cross section of a PCB transmission path, which illustrates many discontinuities.

Figure 50. Example TDR Voltage Plot for a Cross Section of a PCB



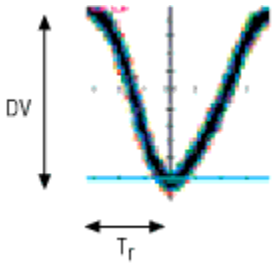
If you experience a TDR plot similar to Figure 51 on page 46, calculate the capacitive discontinuity introduced by the SMA connector by factoring in the voltage dip.

Figure 51. TDR Plot for a Section of a PCB



You can determine Tr and V for the equation from the curve as shown in Figure 52 on page 46.

Figure 52. TDR Plot for the SMA



In this example,

From the equation:

The examples in this section can be used when modeling discontinuity with a simulator. However, instead of using TDR to extract the parasitic for the discontinuity, you can model the discontinuity in the 2D and 3D field solvers.

Reflection coefficient = $(Z_{LOAD} - Z_0)/(Z_{LOAD} + Z_0)$

Reflection coefficient in this case = (ɑ– 50)/(ɑ + 50) = 1

$Z_0 = \sqrt{(L/C)}$

$R = Z_0/2$

$RC = Z_0C/2$

$(\Delta V/250\ mV) = 1 - (Tr\ /2RC)$

$R = 2Z_0$

$L/R = L/2Z_0$

$(\Delta V/250\ mV) = 1 - (Tr\ x\ Z_0\ /L)$

$(\Delta V/250\ mV) = 1 - (Tr\ /2RC)$

$RC = (Tr\ x\ 250\ mV)/2\ (250\ mV - \Delta V) = 29.9\ ps$

$RC = Z_0C/2$

If $Z_0 = 50\ \Omega$, then $C = 1.196\ pF$

5.1.5.6. Discontinuities Related to a Transmission Path

Discontinuity on a transmission path degrades signals. Signals with fast rise times have higher degradation than signals with slow rise times. Thus, high-speed board designs require careful planning to avoid the problems associated with discontinuity. This section discusses inductive and capacitive discontinuities related to a transmission path.

5.1.5.6.1. Inductive Discontinuity

Figure 53 on page 47 illustrates a TDR voltage plot for two different SMA connectors, one side of the SMA connector is 50 Ω and the other is 58 Ω. The curve rises upwards due to the increasing inductance in the region.

The two plots in Figure 53 on page 47 represent two different discontinuities due to SMA connectors. The curve with the higher peak represents a connector with higher inductive discontinuity of about 3.8 nH. The curve with a lower peak represents a connector with lower inductive discontinuity of about 2.6 nH. You can calculate the inductance for the discontinuity for both these curves from the graph.

Figure 53. Impedance Curves for SMA Connectors

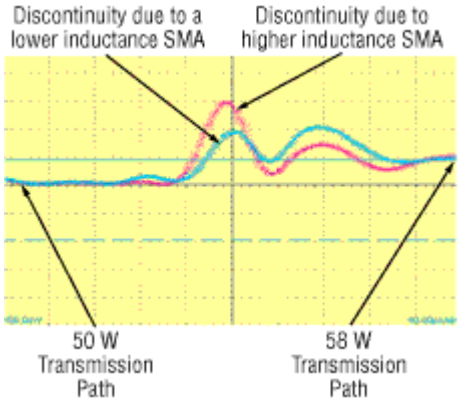
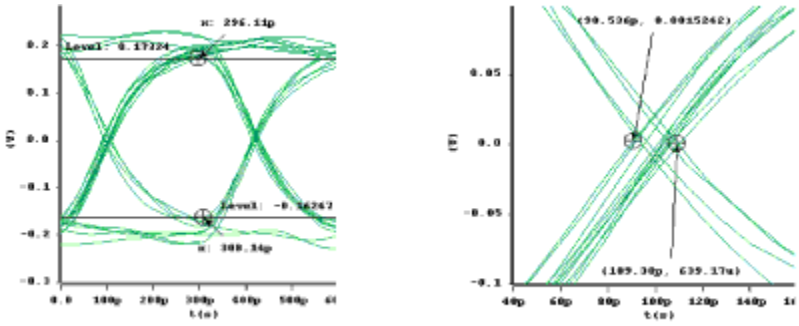


Figure 53 on page 47 shows a 3.125-Gbps signal transmitted through the two SMA connectors. The rise time of the signal is approximately 70 ps.

Figure 54 on page 48 shows the eye opening plot when a signal passes through the lower-inductance (2.6 nH of discontinuity) SMA connector. The eye opening is 336 mV, and the jitter is 20 ps.

Figure 54. Lower-Inductance SMA Connector Eye Opening and Expanded View of the Eye Opening



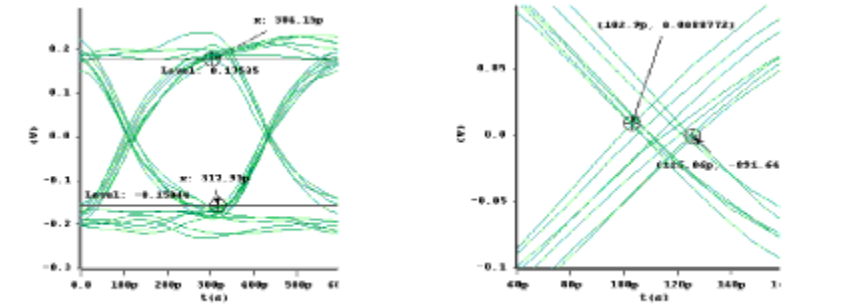
Eye Opening, Lower-Inductance SMA Connector *Expanded View, Lower-Inductance SMA Connector*

An expanded view of the eye (Figure 54 on page 48) provides a better jitter reading; the peak-to-peak jitter value is approximately 20 ps.

Figure 55 on page 48 shows an eye opening plot of the same signal; however, this time the signal goes through 3.8 nH of inductive discontinuity due to the higher-inductance SMA connector. The eye opening is approximately 332 mV. When comparing the plots, the plot in Figure 55 on page 48 has more jitter than Figure 54 on page 48.

An expanded view of the eye (Figure 55 on page 48) provides a better jitter reading; the peak-to-peak jitter value is approximately 24 ps.

Figure 55. Higher-Inductance SMA Connector Eye Opening and Expanded View of the Eye Opening



Eye Opening, Higher-Inductance SMA Connector *Expanded View, Higher-Inductance SMA Connector*

Jitter increases and the eye opening gets smaller when the wrong type of connectors are used or other forms of inductive discontinuities are added to the transmission path. Increasing jitter behavior becomes a significant problem with signals with faster rise times. Also, when the signals become more stressed (i.e., random), jitter is more pronounced.

5.1.5.6.2. Capacitive Discontinuity

This section discusses the effects of capacitive discontinuity, which usually occurs when components are introduced on the transmission path.

The two connector plots in Figure 56 on page 49 show capacitive loads, one acting as a lower capacitive discontinuity and the other as higher capacitive discontinuity. The capacitance (C) for the load can be calculated with the equation:

A 3.125-Gbps signal (a pseudo random binary sequence (PRBS) pattern) is sent through the first connector that looks like a lower-capacitive connector (1.2 pF); the eye opening and jitter are observed on the other end.

Figure 56. Lower- and Higher-Capacitive Load Connectors Illustrating the Effects of Capacitive Discontinuity

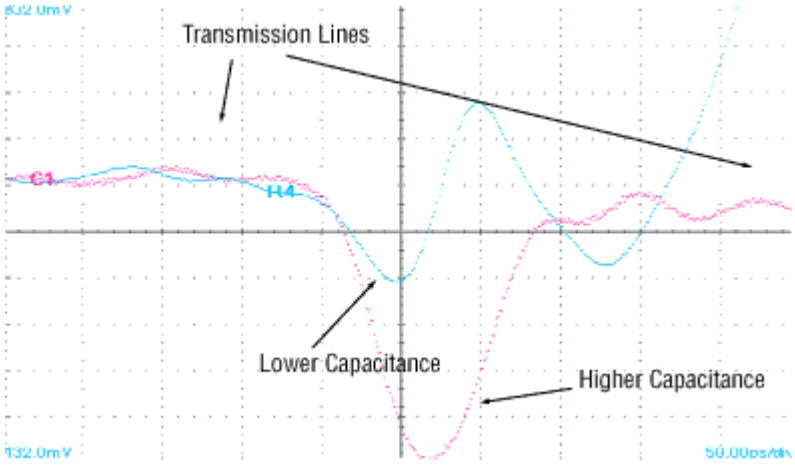
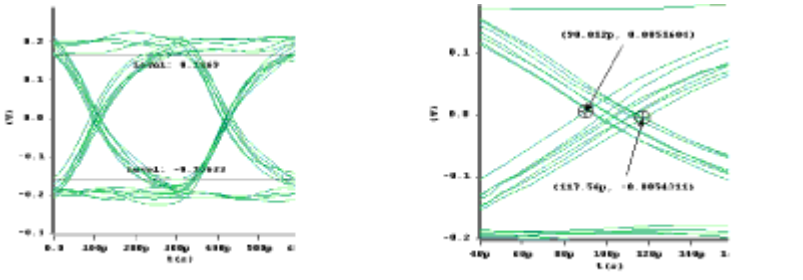


Figure 57 on page 49 shows the eye opening with the connector that induces a discontinuity of 1.2 pF. The eye opening is a 330-mV differential. The expanded view of the eye shows the peak-to-peak jitter as approximately 27 ps.

Figure 57. Lower-Capacitance Connector Eye Opening and Expanded View of the Eye Opening

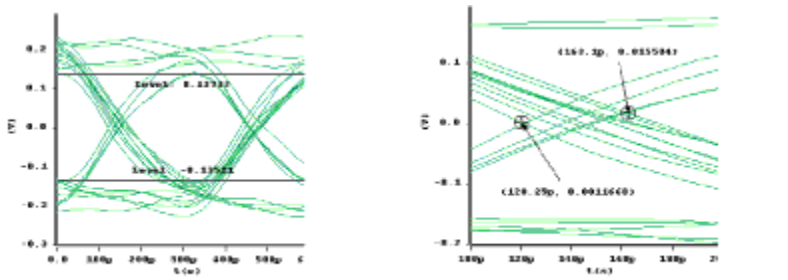


Eye Opening, Lower-Capacitance Connector *Expanded View, Lower-Capacitance Connector*

Figure 57 on page 49 shows a 3.125-Gbps PRBS pattern sent through the second connector that looks like a higher-capacitive connector; the eye opening and jitter are observed on the other end.

Figure 58 on page 50 shows the eye opening for the same signal passing through an SMA connector with a capacitance of 2.9 pF. The eye opening is approximately 280 mV, differential. The expanded view of the eye opening shows that the peak-to-peak jitter is 43 ps.

Figure 58. Higher-Capacitance Connector Eye Opening and Expanded View of the Eye Opening



Eye Opening, Higher-Capacitance Connector *Expanded View, Higher-Capacitance Connector*

You should avoid adding connectors and components on the transmission path whenever possible. However, if connectors are required, select ones that creates the least amount of inductive and/or capacitive discontinuity on the transmission path. The jitter and amplitude impact on a 3.125-Gbps signal when transmitting through a 2.9 and 1.2 pF capacitor is very significant. The eye opening shows an amplitude difference of 50 mV, and the expanded view shows a peak-to-peak jitter difference of 16 ps.

- $\rho = RC = (Z_0C/2)$

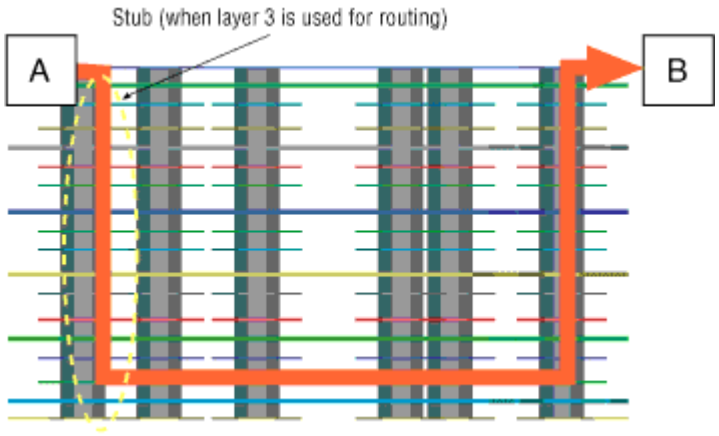
5.1.5.6.3. Via Discontinuity

Avoid vias and layer changes as much as possible when routing a trace because vias slow down edges and cause reflections. Vias are both inductive and capacitive in nature; however, they are dominantly capacitive. A design that uses differential signals requires vias. However, to ensure that the true and complement signals experience the same discontinuity, vias must be in the same configuration for each signal of the differential pair. Thus, any variation in signal due to the via-induced discontinuity is in a common mode. A differential mode discontinuity causes a reduction in the dynamic range.

Blind vias are more expensive, smaller, and act less as a discontinuity than full-sized vias. Blind vias do not go through the PCB and are designed to reduce discontinuity from vias. For better performance when using full-sized vias, use vias in series with the transmission line. The via section that is left hanging behaves like a capacitive stub.

Figure 59 on page 51 shows an 18-layer board. Layers 1, 3, and 16 are signal layers. Route a trace from layer 1 down to layer 16, rather than routing through layer 3. If you route a trace that stops at layer 3, then the part of the via left hanging behaves like a capacitive stub.

Figure 59. Eighteen-Layer Board with Trace Stub



The capacitive stub effects on a via become more pronounced when the board design involves:

A board thickness of 93 mils, with capacitive stubs, has less impact on a 3.125-Gbps signal as compared to a 200 mils-thick board running at the same frequency. Thus, vias affect signal integrity (at 3.125 Gbps) for boards that are too thick.

When possible, avoid vias and via stubs, and remove any unnecessary pads on vias because the pads create parallel plate capacitance between each other. When designing a 100-mils-thick board, you do not need to back drill the vias for a 3.125-Gbps signal. However, back drilling may be advisable for boards measurably thicker than 100 mils.

A current flow on a transmission line creates a magnetic field. The flux lines induce a return current on the reference structure. When a transmission line has its broadside facing reference planes, most of the return current travels underneath the transmission line at a skin depth on the reference plane. The value of skin depth can be calculated with the following equation:

Where:

You can calculate the current density at any point x in the reference plane with the following equation:

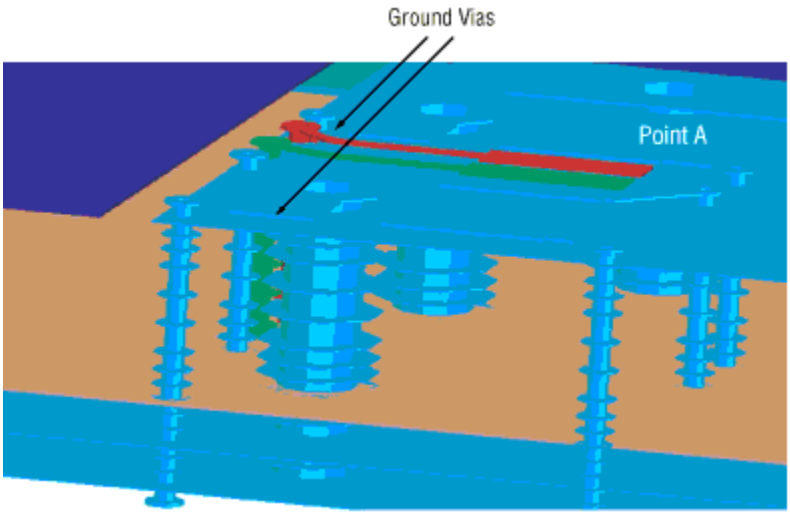
Where:

You should provide a good path for return currents. Figure 60 on page 52 shows a layer change (from layer 1 to 13) for a pair of differential signals (i.e., red and green structures). The signal starts at point A (in Figure 60 on page 52) and transmits to point B (Figure 62 on page 53).

Figure 60 on page 52, Figure 61 on page 52, and Figure 62 on page 53 show that solid reference planes (i.e., light blue structures) are provided for the signal lines.

Create GND islands when necessary. When creating islands of GND, ensure that other signals referencing the plane do not pass over the split. If a signal does pass over the split, its loop increases, also increasing the inductance in the region.

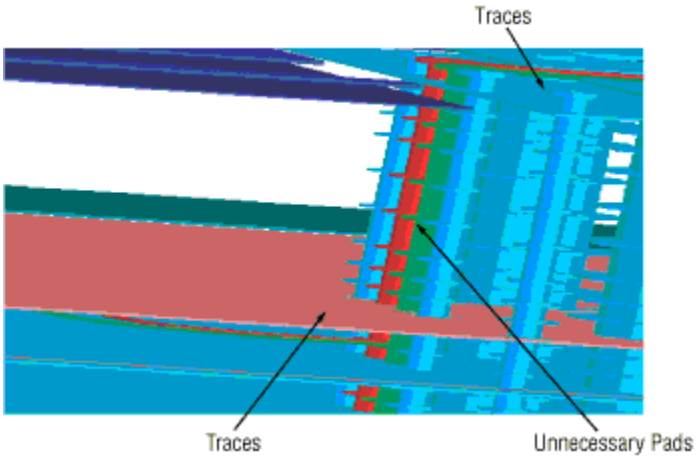
Figure 60. Layer Changes



At the point of the layer change, GND vias should be provided for the return current paths. If the return path does not have GND vias, the return currents look for the closest path, but these paths may not be close enough. In this scenario, the current takes a longer path, increasing its loop. Because of the number of flux lines going through the loop, increasing the loop also increases the inductance. Although Figure 60 on page 52 only shows two vias, it is better to have more vias circling the signal vias.

Figure 61 on page 52 is a side view of the layer change view in Figure 60 on page 52. The signals transmit from layer 1 to the layer 13. Each layer has via pads. Because there is parallel plate capacitance between the pads, the unnecessary pads add capacitive loading. Therefore, remove all of the pads except the ones that directly connect the via to the transmission lines.

Figure 61. Side View of Layer Changes



In Figure 62 on page 53, a GND island is provided to give a good reference path for the signal. GND vias (i.e., the light blue structures) are brought up to avoid too much discontinuity.

The PCB in Figure 62 on page 53 does not have enough GND vias, so you should add more around the signal vias, evenly distributed for the two signal lines. In Figure 62 on page 53, only one side of the differential pair has a GND via close to it.

Figure 62. Transmission Path to Point B

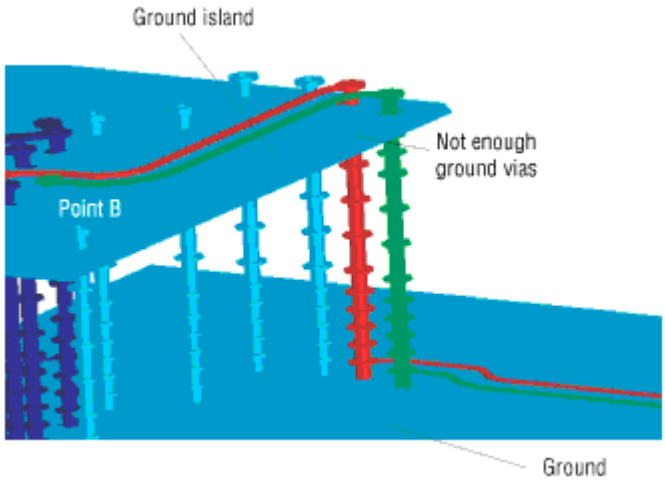
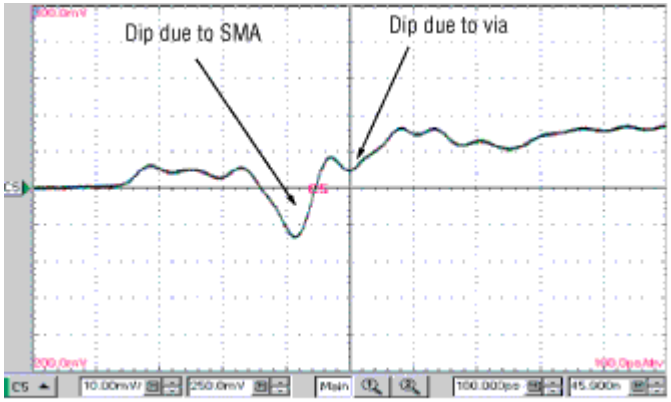


Figure 63 on page 53 shows a TDR plot that contains an example via from the Stratix GX development board, a 93-mils thick board. The via looks like a capacitive discontinuity of 0.7 pF. The via connects two transmission lines that are on layer 1 and layer 13 of an 18-layer board.

Figure 63. Capacitive Discontinuity Due to a Via on a 93-mil Thick Board



- Higher signal speeds
- Thicker boards
- Non-essential extra via pads
- skin depth = $1/\sqrt{(\pi f \mu_o \mu_r \sigma)}$
- f = frequency
- μo = magnetic permeability of air
- μr = relative magnetic permeability
- σ = conductivity of the material

- $Ix = I_o e^{-x/do}$
- Ix = current density at x
- Io = current density on skin depth
- x = distance from surface
- do = skin depth

5.1.5.6.4. Right-Angle Bends Related to a Transmission Path

To minimize impedance discontinuities on the transmission line, avoid using right-angle bends. At the bend, the effective transmission line width increases, this result in an impedance discontinuity, increasing the capacitance.

Instead of 90° bends, use mitered 45° bends. Mitered 45° bends reduce reflection on the signal by minimizing impedance discontinuities. Right-angled bends also look like antennas. Figure 64 on page 54 shows a 60-mils transmission line immersed in FR-4 dielectric (εr = 4.1, loss tangent = 0.022) with dimensions for 50-Ω impedance. The 90° and 45° bend (refer to Figure 65 on page 55) traces are simulated using SPICE models. The parasitics are extracted with a 3D field solver.

Figure 64. 90° Bend on a Transmission Line

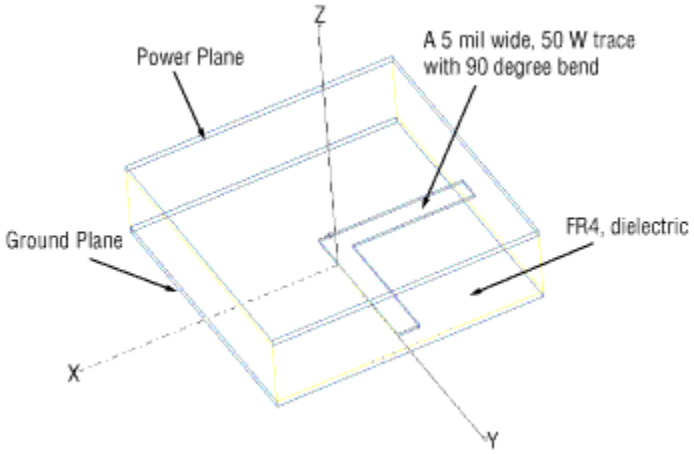


Figure 65. 45° Bend on a Transmission Line

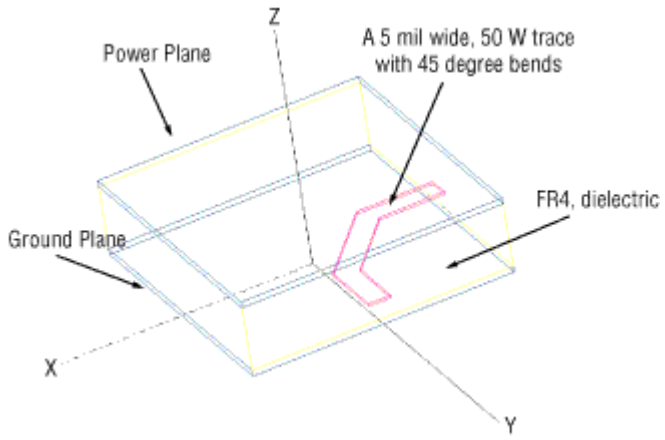
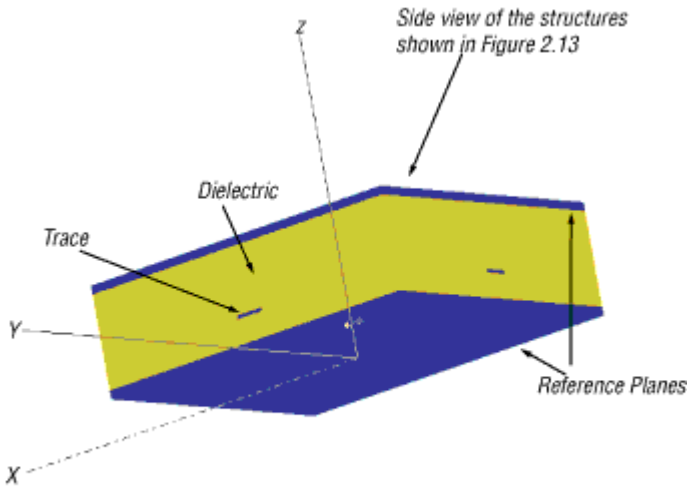


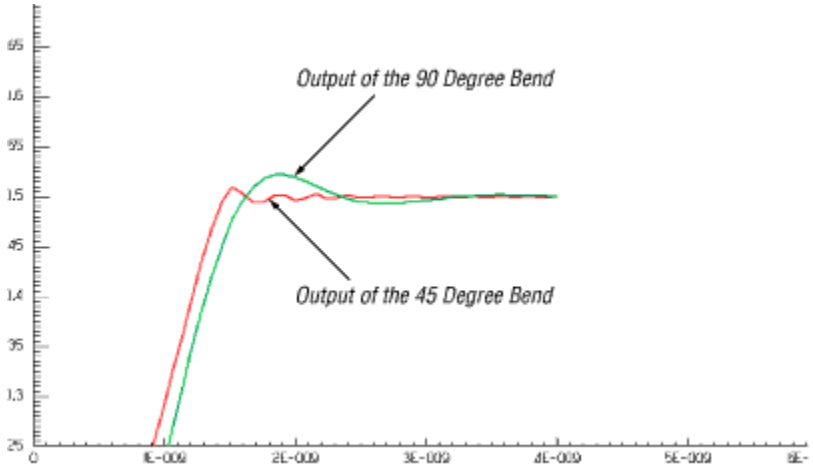
Figure 66 on page 55 shows the board's cross section.

Figure 66. Board's Cross Section



A 1-ns (rise time) signal is fed to one side of the trace and the output is observed at the other end. Because of the extra capacitive loading, the output of the 90° bend has a slight delay and more ringing on it. When driving through long traces or other stressful conditions, even a little bit of ringing is destructive. For instance, adding more closure to an almost closed eye can result in the receiver failing to recognize some bits of data. The 90° bend affects signals running at 3.125 Gbps even more severely. Figure 67 on page 56 shows the effects of bends on signals.

Figure 67. Effects of Bends on Signals



5.1.6. Ground Bounce

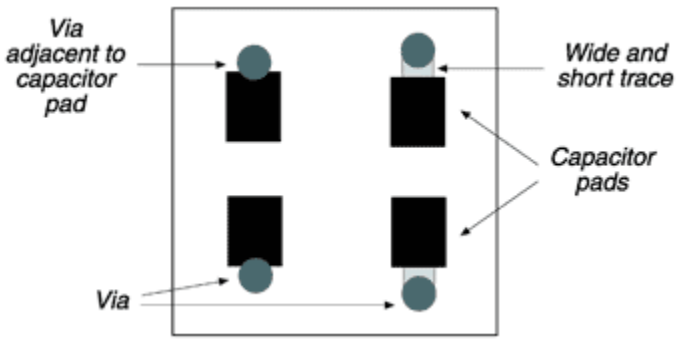
As digital devices become faster, their output switching times decrease. Faster switching times cause higher transient currents in outputs as they discharge load capacitances. These higher currents, which are generated when multiple outputs of a device switch simultaneously from a logic high to a logic low, can cause a board-level phenomenon known as ground bounce.

Many factors contribute to ground bounce. Therefore, no standard test method predicts ground bounce magnitude for all possible PCB environments. Determine each condition's and each device's relative contributions to ground bounce by testing the device under these conditions: Load capacitance, socket inductance, and the number of switching outputs, which are the predominant conditions that influence the magnitude of ground bounce in programmable logic devices (PLDs).

5.1.6.1. Design Guidelines

It is recommended to follow the below design methods to reduce ground bounce:

Figure 68. Suggested Via Location that Connects to Capacitor Pad



These design guidelines provide information and help for high-speed logic designs operating over a range of PCB conditions.

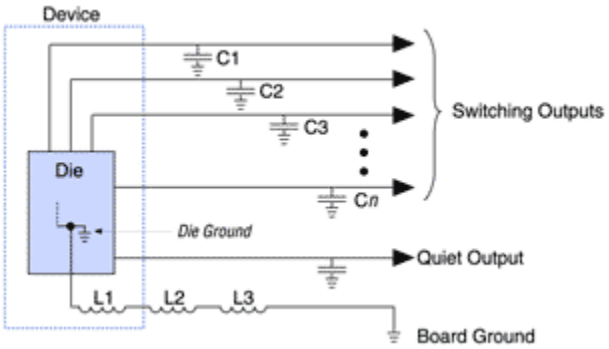
- Add decoupling capacitors for as many V_{CC}/GND pairs as possible.
- Place the decoupling capacitors as close as possible to the power and ground pins of the device.
- Add external buffers at the output of a counter to minimize the loading on Intel® device pins.
- Configure the unused I/O pin as an output pin and then drive the output low. This configuration acts as a virtual ground. Connect this low driving output pin to GNDINT and/or the board's ground plane.
- Any unused I/O pin may be driven to ground by programming the “programmable ground” bit (one per I/O cell). In doing so, the macrocell does not need to be sacrificed, but can be used as a buried macrocell.
- When speed is not critical, turn on the slow slew rate logic option.
- Limit load capacitance by buffering loads with an external device, or by reducing the number of devices that drive the bus.
- Eliminate sockets whenever possible.
- Reduce the number of outputs that can switch simultaneously and/or distribute them evenly throughout the device.
- Move switching outputs close to a package ground pin.
- Create a programmable ground next to switching pins.
- Eliminate pull-up resistors or use pull-down resistors.
- Use multi-layer PCBs that provide separate V_{CC} and ground planes.
- Add appropriate resistors in series to each of the switching outputs to limit the current flow into each of the outputs.
- Create synchronous designs that are not affected by momentarily switching pins.
- Assign I/O pins to minimize local bunching of output pins.
- Place the power and ground pins next to each other. The total inductance is reduced by mutual inductance, because current flows in opposite directions in power and ground pins.
- Use a bigger via size to connect the capacitor pad to the power and ground plane to minimize the inductance in decoupling capacitors.
- Use the wide and short trace between the via and the capacitor pad or place the via adjacent to the capacitor pad. Refer to Figure 68 on page 56.
- Use surface mount capacitors to minimize the lead inductance.
- Use low effective series resistance (ESR) capacitors. The ESR should be < 400 mÎ©.
- Each GND pin/via should be connected to the ground plane individually.
- To add extra capacitance on the board, Intel recommends placing a ground plane next to each power (V_{CC}) plane. This placement gives zero lead inductance and no ESR. The dielectric thickness between the two planes should be ~5 mils.

5.1.6.2. Analyzing Ground Bounce

Figure 69 on page 58 shows a simple model for analyzing ground bounce. The external components driven by the device appear as capacitance loads to that device (C1 to Cn). These capacitive loads store a charge determined by the following equation:

Thus, the charge increases as the voltage and/or load capacitance increases.

Figure 69. Ground Bounce Model



A device's environment and ground path have intrinsic inductances (shown in Figure 69 on page 58 as L1, L2, and L3). L1 is the inductance of the bond wire from the device's die to its package pin, and of the pin itself. L2 is the inductance of the connection mechanism between the device's ground pin and the PCB. This inductance is greatest when the device is connected to the PCB through a socket. L3 is the inductance of the PCB trace between the device and the PCB location where the power supply's reference ground is connected.

Ground bounce occurs when multiple outputs switch from high to low. The transition causes the charge stored in the load capacitance to flow into the device. The sudden rush of current (di/dt) exits the device through the inductances (L) to board ground, generating a voltage (V) determined by the equation V = L × (di/dt). This voltage difference between board ground and device ground causes the relative ground level for low or quiet outputs to temporarily rise or bounce. Although the rush of current is brief, the magnitude of the bounce can be large enough to trigger other devices on the PCB.

In synchronous designs, ground bounce is less often a problem because synchronous outputs have enough time to settle before the next clock edge. Also, synchronous circuits are not as likely to be falsely triggered by a voltage spike on a quiet output.

Capacitive loading on the switching outputs and quiet outputs affect ground bounce differently.

- Charge (Q) = [voltage (V) × capacitance (C)]

5.1.6.3. Switching Outputs

When the capacitive loading on the switching outputs increases, the amount of charge available for instantaneous switching increases, which in turn increases the magnitude of ground bounce. Depending on the device, ground bounce increases with capacitive

loading until the loading is approximately 100 pF per device output. At this point, the device output buffers reach their maximum current-carrying capacity and inductive factors become dominant.

One method of reducing the capacitive load and, consequently, ground bounce is to connect the device's switching outputs to a bus driver integrated circuit (IC). The outputs of this IC drive the heavy capacitive loads, reducing the loading on the device and minimizing ground bounce for the device's quiet outputs.

Some bus applications use pull-up resistors to create a default high value for the bus. These resistors cause the load capacitances to charge up to the maximum voltage. Consequently, the driving device produces a higher level of ground bounce. Eliminate pull-up resistors in applications in which ground bounce is a concern, or design bus logic that uses pull-down resistors instead.

The number of switching outputs also affects ground bounce. As the number of switching outputs increases, the total charge stored also increases. The total charge is equal to the sum of the stored charges for each switching output. Therefore, the amount of current that must sink to ground increases as the number of switching outputs increases. Ground bounce can increase by as much as 40 to 50 mV for each additional output that is switching.

To counteract these effects, Intel devices provide multiple VCC and GND pin pairs. Reduce ground bounce by moving switching outputs close to a ground pin and distributing simultaneously switching outputs throughout the device.

Besides placing switching pins next to a ground pin, create a programmable ground pin by creating an output pin in the design that drives only ground. By connecting this output pin to ground on the board, the device ground has another connection to the board ground, which helps reduce ground bounce.

Many Intel FPGA devices have slew rate options for the output drivers. Turning on the slow slew rate option for all or most of the drivers slows down the drivers, decreasing di/dt and reducing ground bounce.

To further reduce ground bounce, limit the number of outputs that can switch simultaneously in the design. For functions such as counters, use Gray coding as an alternative to standard sequential binary coding, because only one bit switches at a time.

In extreme cases, adding resistors in series to each of the switching outputs in a high-speed logic device can limit the current flow into each of the outputs and, thus, reduce ground bounce to an acceptable level.

5.1.6.4. Quiet Outputs

An increase in capacitive loading on quiet outputs acts as a low-pass filter and tends to dampen ground bounce. Capacitive loading on a quiet output can reduce ground bounce by as much as 200 to 300 mV. However, an increase in capacitive loading on a quiet output can increase the noise seen on other quiet outputs when the capacitive-loaded pin does switch.

5.1.6.5. Minimizing Lead Inductance

Using multi-layer PCBs that provide separate VCC and ground planes can also reduce the ground bounce caused by PCB trace inductance. Wirewrapping the VCC and ground supplies usually increases the amount of ground bounce. To reduce unwanted inductance, use low-inductance bypass capacitors between the VCC supply pins and the board ground plane, as close to the package supply pins as possible. It is required to use low ESR decoupling surface mount capacitors of 0.01 µF to 0.1 µF in parallel to reduce ground bounce. Adding a 0.001 µF capacitor in parallel to these capacitors filters high frequency noise (>100 MHz).

For more information see the *Minimizing Ground Bounce & VCC Sag* white paper (PDF).

5.1.7. Understanding Transmission Lines

The transmission line is a trace and has a distributed mixture of resistance (R), inductance (L), and capacitance (C). There are two types of transmission line layouts:

Figure 70 on page 60 shows a microstrip layout, which refers to a trace routed as the top or bottom layer of a PCB and has only one voltage-reference plane (i.e., power or GND). Figure 71 on page 60 shows a stripline layout, which uses a trace routed on the inside layer of a PCB and has two voltage-reference planes (i.e., power and/or GND).

Figure 70. Microstrip Transmission Line Layout

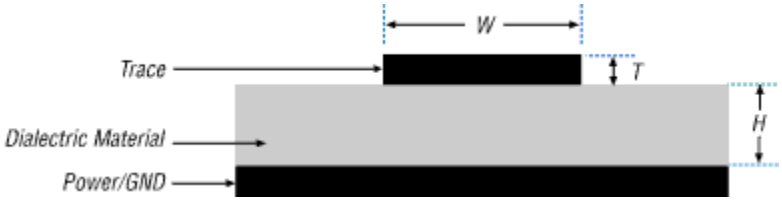
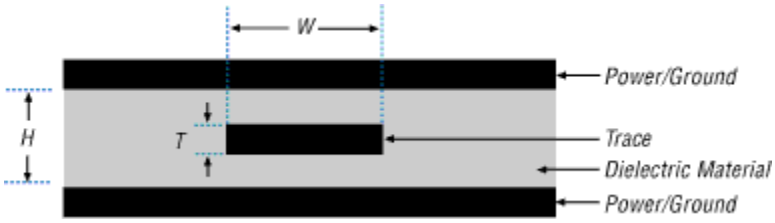


Figure 71. Stripline Transmission Line Layout



Notes to Figure 70 on page 60 and Figure 71 on page 60:

- Microstrip
- Stripline
- W = width of trace, T = thickness of trace, and H = height between trace and reference plane.
- W = width of trace, T = thickness of trace, and H = height between trace and two reference planes.

5.1.8. Impedance Calculation

Any circuit trace on the PCB has characteristic impedance associated with it. This impedance is dependent on the width (W) of the trace, thickness (T) of the trace, dielectric constant (εr) of the material used, and height (H) between the trace and reference plane.

5.1.8.1. Microstrip Impedance

A circuit trace routed on an outside layer of the PCB with a reference plane (i.e., GND or VCC) below it constitutes a microstrip layout. Use the following equation to calculate the impedance of a microstrip trace layout.

Using typical values of W = 8 mil, H = 5 mil, T = 1.4 mil, εr and (FR-4) = 4.1 and solving for microstrip impedance (Z_0) yields:

The measurement unit in the equation is mils (i.e., 1 mil = .001 inches). Also, copper (Cu) trace thickness (T) is usually measured in ounces (i.e., 1 oz = 1.4 mil).

Figure 72 on page 61 shows microstrip trace impedance versus trace width (W) using the values in the equation, keeping dielectric height and trace thickness constant.

Figure 72. Microstrip Trace Impedance with Changing Trace Width

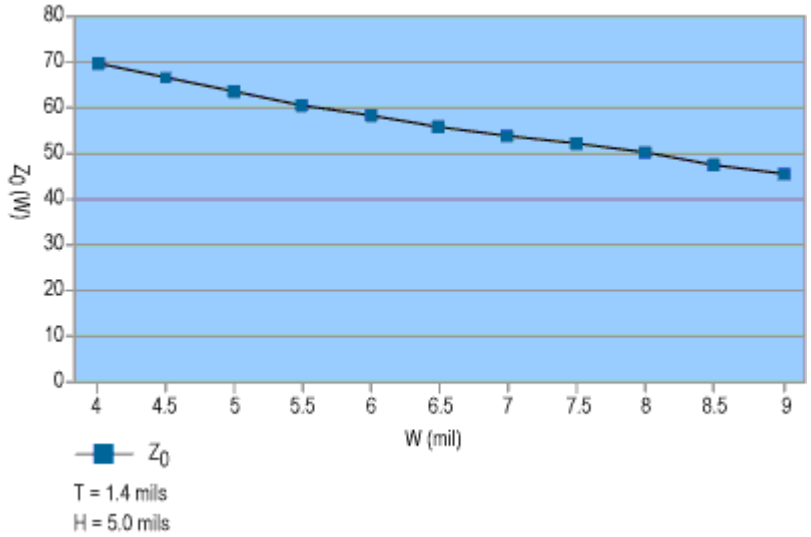
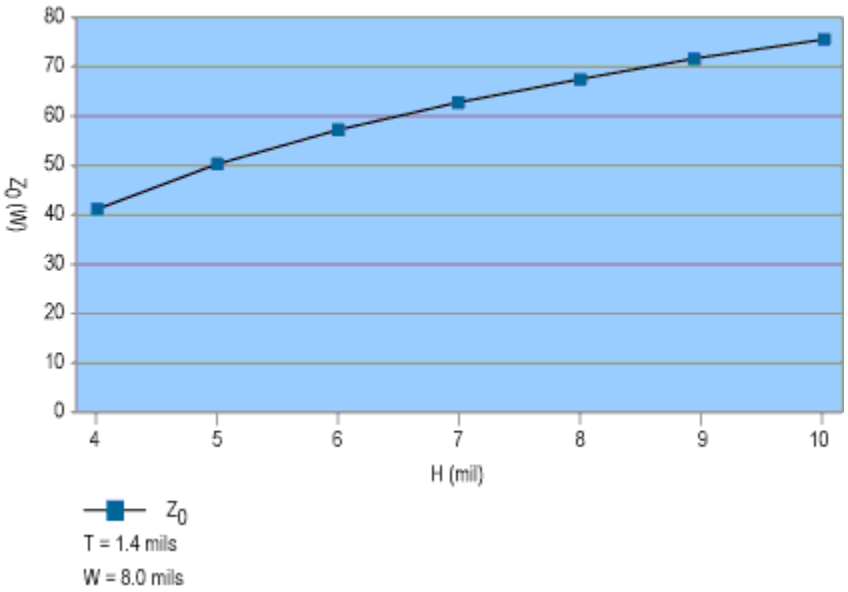


Figure 73 on page 62 shows microstrip trace impedance versus height (H), using the values in the microstrip trace impedance equation, keeping trace width and trace thickness constant.

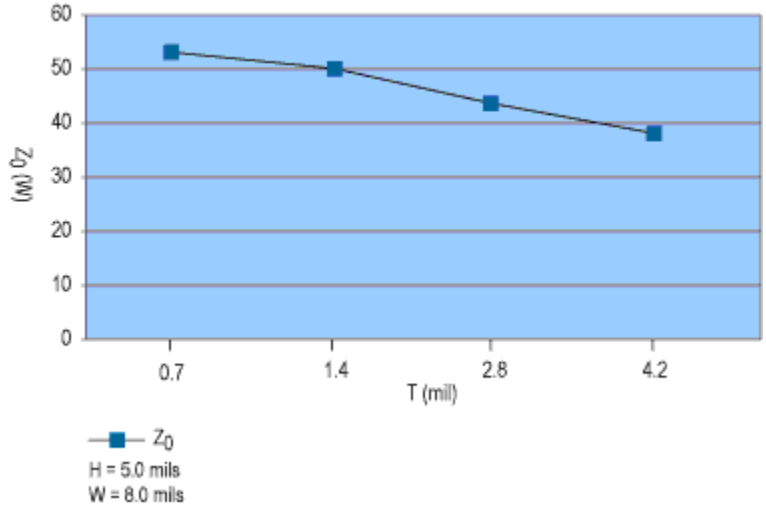
Figure 73. Microstrip Trace Impedance with Changing Height



The impedance graphs show that the change in impedance is inversely proportional to trace width and directly proportional to trace height above the ground plane.

Figure 74 on page 62 plots microstrip trace impedance versus trace thickness (T) using the values in the microstrip trace impedance equation, keeping trace width and dielectric height constant. This figure shows that as trace thickness increases, trace impedance decreases.

Figure 74. Microstrip Trace Impedance with Changing Trace Thickness



- $Z_0 = [(87/\sqrt{(\varepsilon r + 1.41)}) \ln [(5.98 \times H)/(0.8W + T)]\ \Omega$
- $Z_0 = [(87/\sqrt{(4.1+1.41)}) \ln (5.98\times(5)/0.8(.8)=1.4)]\ \Omega$
- $Z_0 \sim 50\ \Omega$

5.1.8.2. Stripline Impedance

A circuit trace routed on the inside layer of the PCB with two low-voltage reference planes (i.e., power and/or GND) constitutes a stripline layout. Use the following equation to calculate the impedance of a stripline trace layout.

Equation 2:

Using typical values of W= 9 mil, H= 24 mil, T = 1.4 mil, εr and (FR-4) = 4.1 and solving for stripline impedance (Z_o) yields:

Figure 75 on page 63 shows impedance versus trace width using the stripline trace impedance equation, keeping height and thickness constant for stripline trace.

Figure 75. Stripline Trace Impedance with Changing Trace Width

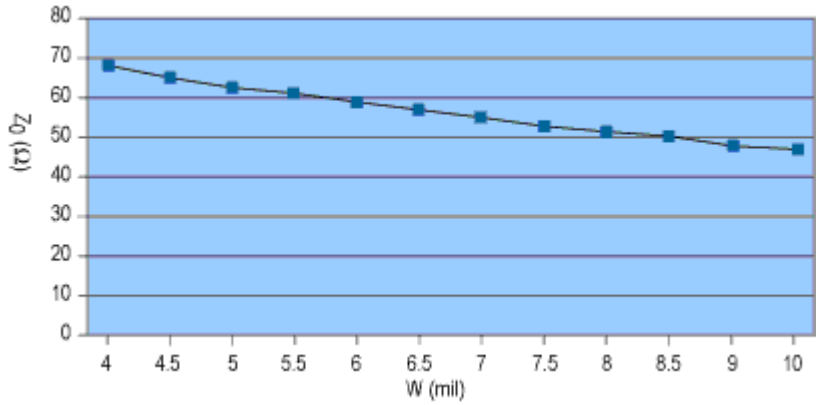
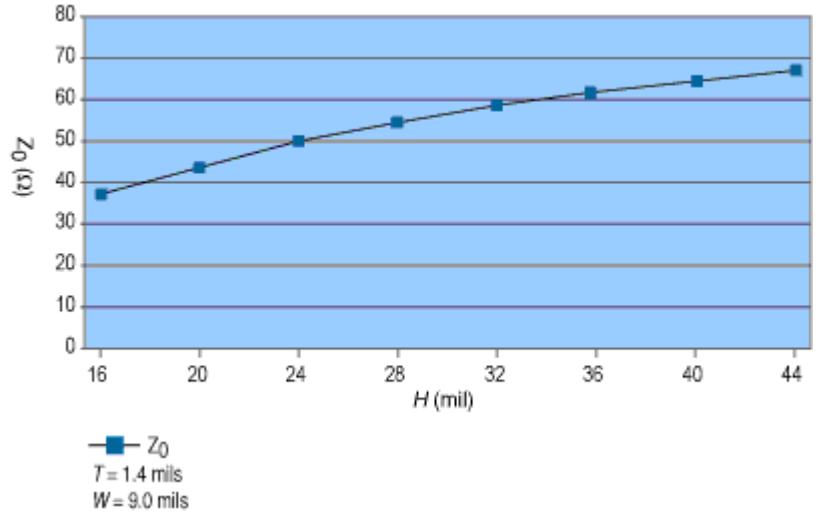


Figure 76 on page 63 shows stripline trace impedance versus dielectric height (H) using the stripline trace impedance equation, keeping trace width and trace thickness constant.

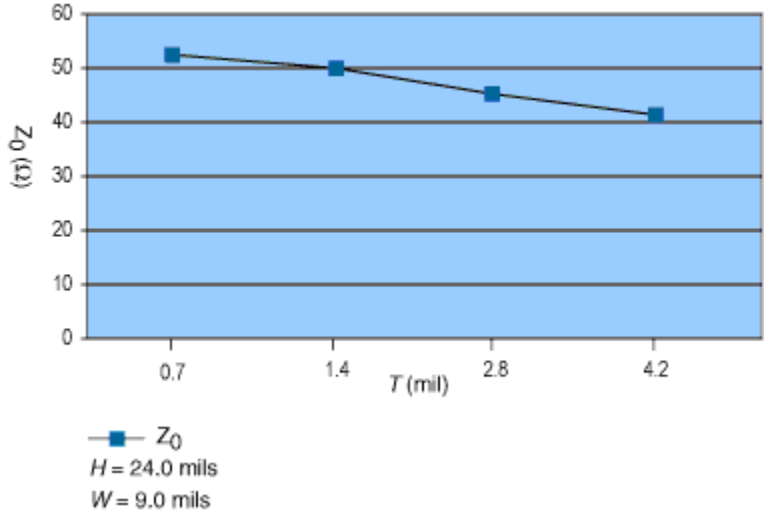
Figure 76. Stripline Trace Impedance with Changing Dielectric Height



As with microstrip layout, the stripline layout impedance also changes inversely proportional to line width and directly proportional to height. However, the rate of change with trace height above GND is much slower in a stripline layout compared to a microstrip layout. A stripline layout has a signal sandwiched by FR-4 material, whereas a microstrip layout has one conductor open to air. This exposure causes a higher, effective dielectric constant stripline layout compared to microstrip layouts. Thus, to achieve the same impedance, the dielectric span must be greater in stripline layouts than in microstrip layouts. Therefore, stripline layout PCBs with controlled impedance lines are thicker than microstrip layout PCBs.

Figure 77 on page 64 shows stripline trace impedance versus trace thickness using Equation 2, keeping trace width and dielectric height constant. This figure shows that the characteristic impedance decreases as the trace thickness increases.

Figure 77. Stripline Trace Impedance with Changing Trace Thickness



- Z_0 = [(60/√εr) ln (4H/0.67π(T+0.8w))] Ω
- Z_0 = [(60/√(4.1)) ln (4(24)/0.67π(1.4)=0.8(9))] Ω
- Z_0 ~50 Ω

5.1.8.3. Propagation Delay

Propagation delay (tPD) is the time required for a signal to travel from one point to another. Transmission line propagation delay is a function of the dielectric constant of the material.

5.1.8.3.1. Microstrip Layout Propagation Delay

Use the following equation to calculate the microstrip trace layout propagation delay.

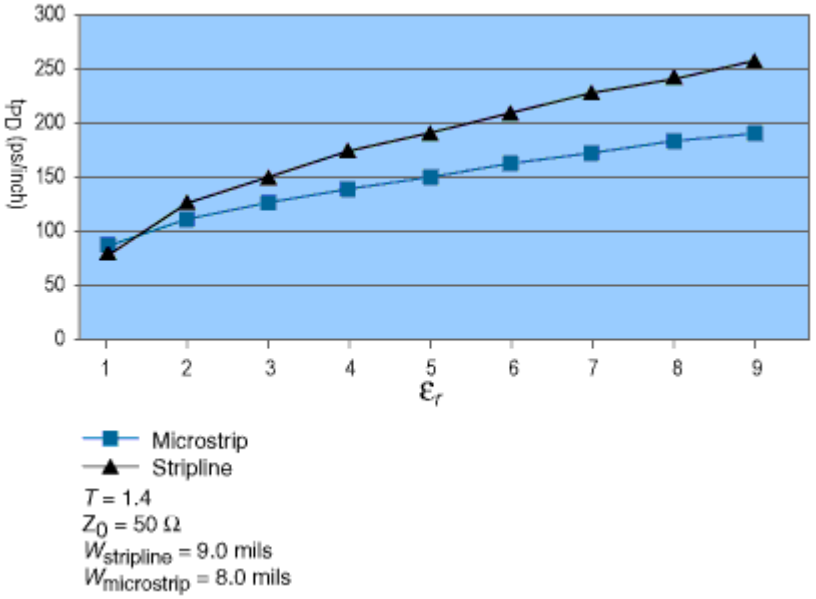
- tPD(microstrip) = 85 √(0.475εr +0.67) ps per inch

5.1.8.3.2. Stripline Layout Propagation Delay

Use the following equation to calculate the stripline trace layout propagation delay.

Figure 78 on page 65 shows the propagation delay versus the dielectric constant for microstrip and stripline traces. As the εr increases, the propagation delay (tPD) also increases.

Figure 78. Propagation Delay vs. Dielectric Constant for Microstrip and Stripline Traces



- tPD(stripline) = 85 √εr ps per inch

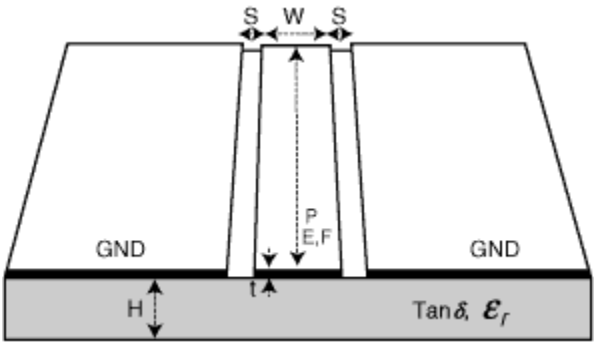
5.1.9. Coplanar Wave Guides

There are three types of coplanar wave guides: simple, grounded, and grounded differential.

5.1.9.1. Simple Coplanar Wave Guide

The simple coplanar wave guide, shown in Figure 79 on page 66, is used primarily in microwave systems. This structure does not require vias, and you can easily mount passive or active devices in the signal path, resulting in a low-loss, high-speed transmission line. The simple coplanar wave guide requires that the substrate thickness (H) be "infinite," so that the fields remain outside the dielectric. You cannot use this structure on multilayer boards because it cannot have a second layer.

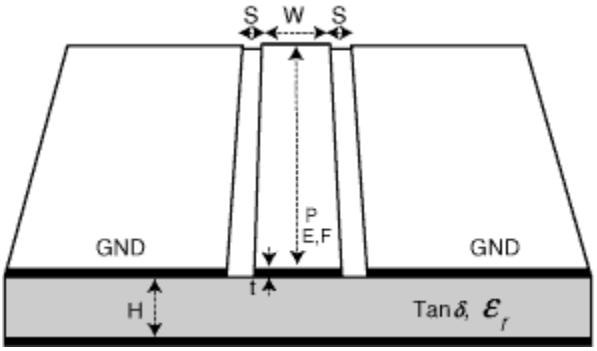
Figure 79. Simple Coplanar Wave Guide



5.1.9.2. Grounded Coplanar Wave Guide

The grounded differential coplanar wave guide is the differential version of the grounded coplanar wave guide and is used in high-speed digital systems that require maximum noise immunity. Figure 80 on page 66 shows the topology. The same limitations for S and G apply in this topology as for the grounded coplanar wave guide.

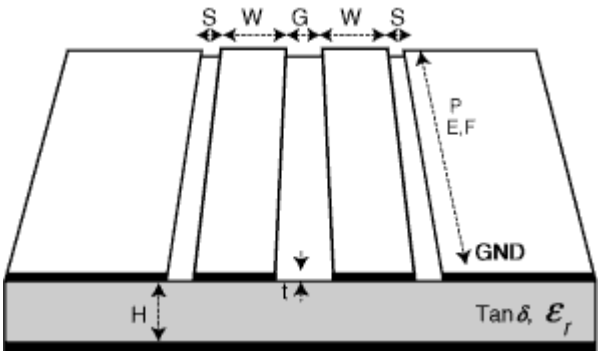
Figure 80. Grounded Coplanar Wave Guide



5.1.9.3. Grounded Differential Coplanar Wave Guide

The grounded differential coplanar wave guide is the differential version of the grounded coplanar wave guide and is used in high-speed digital systems that require maximum noise immunity. Figure 81 on page 67 shows the topology. The same limitations for S and G apply in this topology as for the grounded coplanar wave guide.

Figure 81. Grounded Differential Coplanar Wave Guide

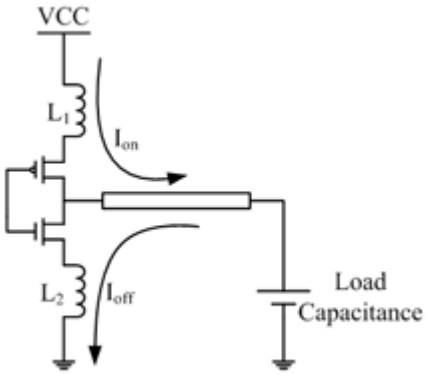


5.1.10. Simultaneous Switching Noise Guidelines

Simultaneous switching noise (SSN) is another important factor to consider when designing a PCB. Although SSN is dominant on the device package, the board layout can help reduce some of the noise.

Every current loop has an inductance value. The current loop in Figure 82 on page 67 has the following inductance:

Figure 82. Inductance Due to a Current Loop



When a driver switches from high to low, a voltage develops in the GND plane, thus:

The noise that develops in the GND plane can become a problem for signal-integrity, especially when there are a lot of drivers switching simultaneously. The noise generated by SSNs can couple to adjacent structures. Proper layout and decoupling reduces noise coupling. The high number of simultaneously switching drivers can cause the power supply to collapse. Thus, the power supply voltage at a certain region loses some of its strength, depending on where the switching is concentrated.

For more information, refer to the following:

- Lloop = L1 (signal) + L2 (GND) – 2 LM (mutual inductance)
- V = Lloop (di/dt)
- *AN 472: Stratix II GX SSN Design Guidelines* (PDF)
- *AN 508: Cyclone III Simultaneous Switching Noise (SSN) Design Guidelines* (PDF)

5.2. Design Security Features in FPGAs

Please refer to the following documents:

- Design Security in Stratix III Devices (PDF) chapter of the Stratix III Device Handbook
- AN 341: Using the Design Security Feature in Stratix II and Stratix II GX Devices (PDF)

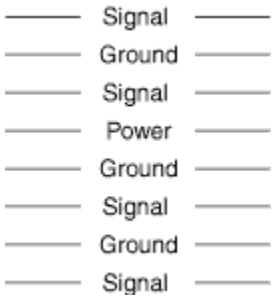
5.3. EMI

Electromagnetic interference (EMI) is directly proportional to the change in current or voltage with respect to time. EMI is also directly proportional to the series inductance of the circuit. Every PCB generates EMI.

Precautions such as minimizing crosstalk, proper grounding, and proper layer stack-up can significantly reduce EMI problems.

Place each signal layer in between the ground plane and power plane. Inductance is directly proportional to the distance an electric charge has to cover from the source of an electric charge to ground. As the distance gets shorter, the inductance becomes smaller as well. Therefore, placing ground planes close to a signal source reduces inductance and helps contain EMI. Figure 83 on page 68 shows an example of an eight-layer stack-up. In the stack-up, the stripline signal layers are the quietest because they are centered by power and GND planes. A solid ground plane next to the power plane creates distribution capacitances with low equivalent series inductance (ESL). With IC edge rates becoming faster, these techniques help to contain EMI.

Figure 83. Example Eight-Layer Stack-Up



Component selection and proper placement on the board is very important to controlling EMI. The following guidelines can help reduce EMI:

- Select low-inductance components, such as surface mount capacitors with low ESR and ESL
- Provide good return path to minimize lop inductance
- Use solid ground planes next to power planes
- Use adjacent ground planes next to each segmented power plane for analog and digital circuits

5.4. Discrete Component Selection for High-Speed Design

Deciding which components to use is an important part of board design. Information and guidelines for selecting discrete components for the PCB is provided below.

5.4.1. Resistors and Capacitors

Choose the smallest footprint available when selecting discrete components such as resistors and capacitors. The small footprint means that the pad on the board can be small and that the parasitic capacitance and inductance will also be small. Intel typically uses 40 mil × 20 mil (0402) package components for high-speed signals.

Inductors typically require bigger footprints because they are often used for power supply filtering and must be bigger to support high currents without saturating.

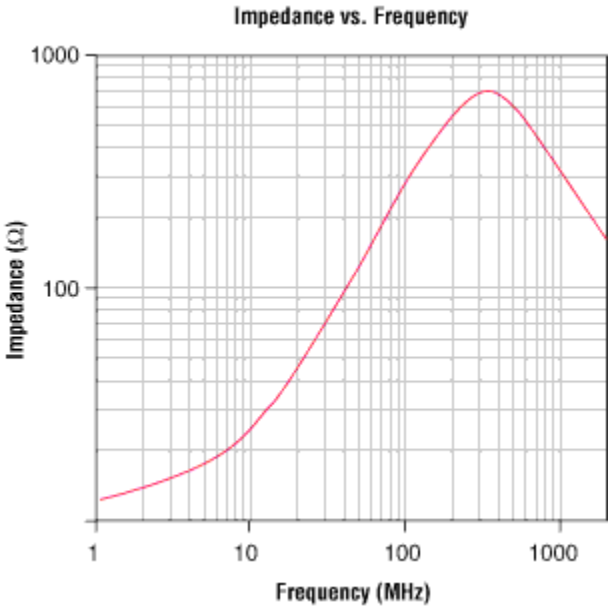
The three parameters of interest when choosing ferrite beads are:

- DC resistance
- AC impedance
- Current handling capability

5.4.2. Inductors and Ferrite Beads

A good ferrite bead has low DC resistance, high AC impedance, and high current handling capability. However, as the current handling capability increases, the AC impedance tends to drop, so there is a tradeoff involved. Figure 84 on page 70 shows the impedance versus frequency plot of a typical ferrite bead.

Figure 84. Typical Impedence Profile of a Ferrite Bead



In Figure 84 on page 70, the impedance at 2 GHz is more than 100 Ω. The ratio between that impedance and the power supply impedance, which is often lower than 1 Ω, is more than 100. As a result, most of the noise is blocked by the ferrite bead and is shunted to ground instead.

Some ferrite beads that meet performance are:

- The Steward MI0805M221R-00 ferrite beads for transceiver power and ground planes. The DC resistance for this part is lower than 50 mΩ, and it can handle 2.5 A of current. The impedance is over 200 Ω at 1 GHz.
- Two Steward ferrite beads connected in parallel can provide 5 A of current capability with 25 mΩ of DC resistance and over 100 Ω of AC impedance. This performance level is adequate for most applications.
- The Murata BLM31PG500SN1 ferrite bead has 25 mΩ of DC resistance, 3 A of current, and 75 Ω of AC impedance at 1 GHz.

5.4.3. SMA Connectors

SMA connectors are typically used for high-speed signals because of their controlled impedance, mechanical robustness, and good signal integrity. These connectors come in the form of edge launch or vertical launch. For edge launch, the connector is connected to the board edge, the central conductor stays flush with the board, and the board is sandwiched between the ground conductors.

The vertical launch type is through-hole or surface mount. For the through-hole type, all the legs and the center conductor go through the board. For a surface mount type, the center conductor barely touches the top layer of the board.

The type of launch technique does not affect the signal quality very much. However, a long center conductor adds inductance to the transmission path and degrades the signal.

With vertical launch surface mount, you can often strip the center conductor down to below 20 mils. This stripping might be slightly harder to achieve in edge launch configuration, but the choice is dependent on the capability of the SMA manufacturers. Intel has stripped the center conductor down to less than 20 mils short in its designs. Intel uses Lighthorse Technologies and Northrop Grumman for SMA connector requirements.

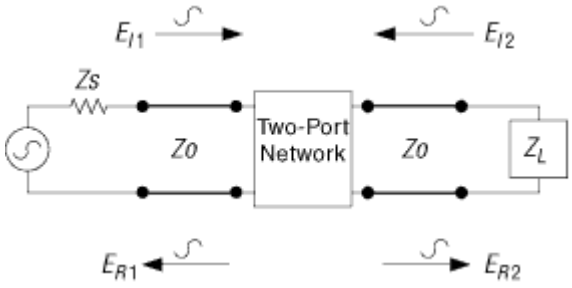
5.5. S-Parameters

It is difficult to define voltages or currents in transmission lines and to measure them at microwave frequencies, because direct measurements usually involve the magnitude (inferred from power) and phase of a wave traveling in a given direction or the standing wave. Thus, equivalent voltages and currents, and the related impedance and admittance matrices, become somewhat of an abstraction when dealing with high-frequency networks.

A representation more consistent with direct measurements, and with the ideas of incident, reflected, and transmitted waves, is provided by the scattering matrix (S-parameters). S-parameters are often used to quantify the impedance, total losses, input return loss, insertion loss, and isolation and crosstalk for the different transmission lines structures. These concepts are most useful at those frequencies where you must consider distributed rather than lumped parameters.

To understand S-parameters, consider the two-port network shown in Figure 85 on page 71. The network can contain a transmission line or any linear time invariant component. The incident voltages at ports 1 and 2 are E_{11} and E_{12}; and the reflected voltages are E_{R1} and E_{R2}. Z_0 is the characteristic impedance of the transmission line. Z_S and Z_L are the source and the load impedances.

Figure 85. S-Parameters



Taking the incident and reflected voltage waves on each side of the two port network and dividing them by the square root of the characteristic impedance, Z_0, results in new variables (a_1, a_2, b_1, and b_2), which are the normalized voltage wave amplitudes at each port.

The square of the magnitude of a_1 ($|a_1|^2$) represents the incident power in port 1, and ($|b_1|^2$) represents the reflected power from this port. The same relations apply to port 2.

The resulting parameters relate the scattered wave (reflected wave) from the network to the incidents waves. These parameters are called S-parameters and are defined by:

Where, S_{11} is the input reflection coefficient, S_{21} is the forward transmission through the network, S_{12} is the reverse transmission through the network, and S_{22} is the output reflection coefficient.

The measurement setup for S-parameters is shown in Figure 86 on page 72 and Figure 87 on page 72. Apply the following conditions for these measurements:

Figure 86. Measurement Setup for S_{11} and S_{21}

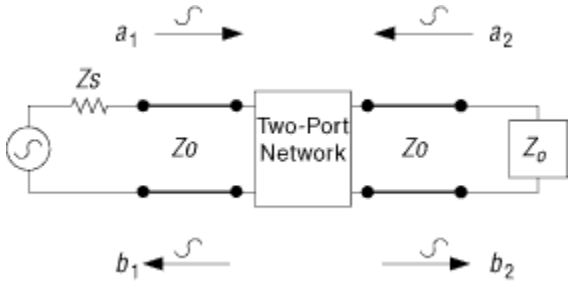
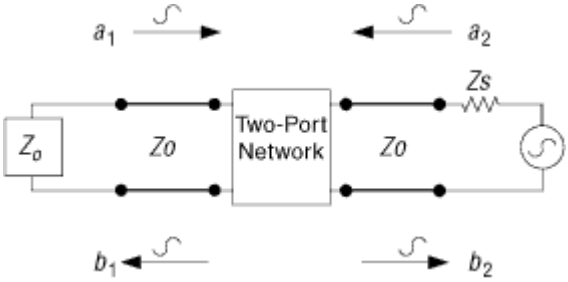


Figure 87. Measurement Setup for S_{22} and S_{12}



For example, to measure S_{11}, ensure that $a_2 = 0$. Do this by making sure that there is no reflection from the load (in other words, set $Z_L = Z_0$).

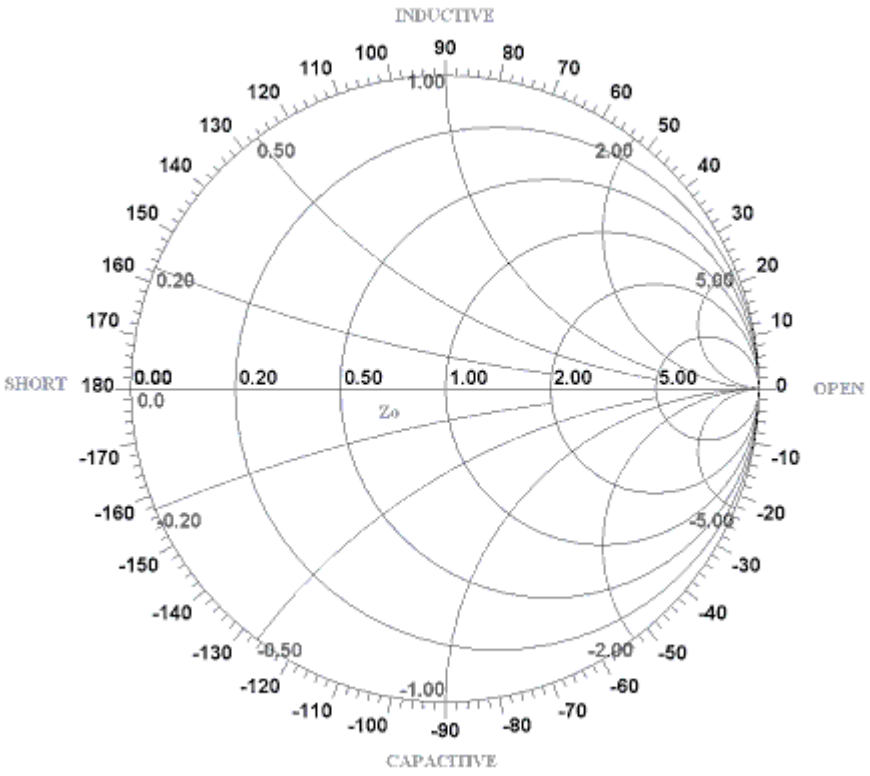
S-parameters are typically measured using network analyzers.

- $a_1=E_I1/\sqrt{Z_0}$ $a_2=EI2/\sqrt{Z_0}$
- $b_1=E_R1/\sqrt{Z_0}$ $b_2=EI2/\sqrt{Z_0}$
- $b_1=S_{11}a_1+S_{12}a_2$
- $b_2=S_{21}a_1+S_{22}a_2$
- $S_{11}=(a_2/b_1) \mid a_2=0$
- $S_{21}=(b_2/a_1) \mid a_2=0$
- $S_{12}=(b_1/a_2) \mid a_1=0$
- $S_{22}=(b_2/a_2) \mid a_1=0$

5.6. Smith Chart

The Smith Chart is a graphical representation of the impedances in a complex plane. You can use this to analyze transmission line impedance matching networks and capacitive or inductive behavior of loads. Figure 88 on page 73 shows a typical Smith Chart. If the impedance is inductive, it shows up in the top half of the circle; if it is capacitive, it shows up in the bottom half. The far-right point in the middle represents an open circuit; the far left represents a short circuit. The middle point (label 1.0) represents a perfect load match to the characteristic impedance of the line.

Figure 88. Smith Chart



5.7. AC Versus DC Coupling

AC coupling refers to the use of a series capacitor on a signal to block the DC signals from going through. DC coupling refers to the case where this capacitor is not present and the signal passes through without any interruption. In AC coupling, a DC restore circuit is generally required after the capacitor to ensure that the common mode voltage requirements of the receiver are met. In Intel devices with transceivers, the DC restore circuitry is built into the device. In that case, external DC restore circuitry is not necessary. DC coupling works only in cases where the output common mode voltage of the transmitter is in the required range of the input common mode voltage of the receiver.

The advantage of AC coupling is that it allows chips with different common mode voltages to interface with each other. The disadvantage is that it requires an extra capacitor, which can add some jitter or other degradation if not properly selected.

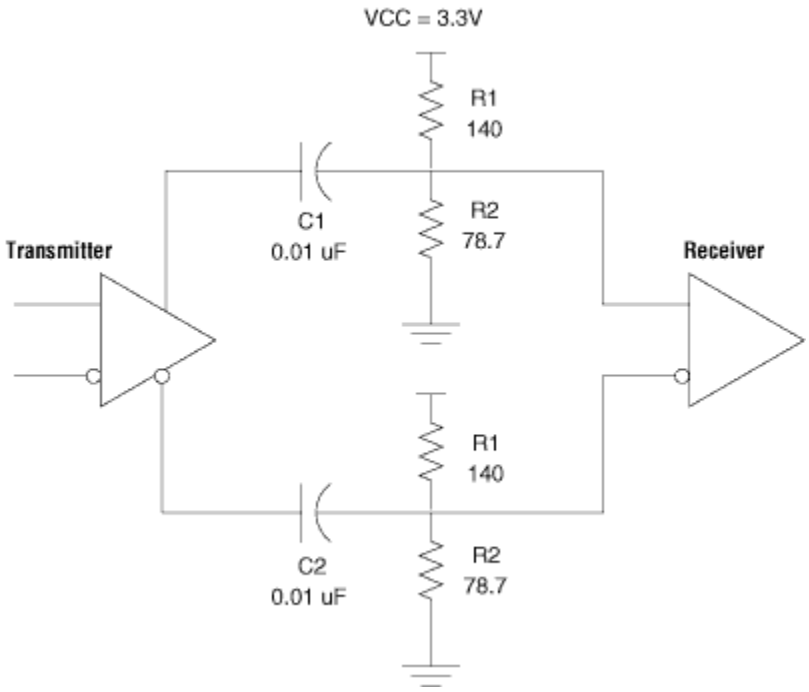
If you are certain that the common mode voltage requirements of the receiver are a subset of the common mode voltage output of the transmitter, use DC coupling. If you are in a borderline case, or if the requirements are not satisfied, then use AC coupling.

In choosing the value of the coupling capacitor, consider what happens if the capacitor is too big or too small. If the capacitor is too big, it can significantly slow the signal down and can also respond poorly to fast changing input signals because of the long charge and discharge times. If the capacitor is too small, it presents a fair amount of impedance and can increase attenuation and change the characteristic impedance of the path. A good balance between these two conflicting requirements is a 0.01 µF capacitor, which Intel uses for its 3.125 Gbps transceiver designs.

When selecting components, use the smallest size possible, because smaller size components have smaller size pads, which reduce the discontinuity. Intel has used 0402 components (40 mils x 20 mils) in its designs.

You can design the DC restore circuitry in a variety of ways. Intel typically uses a simple resistive voltage divider (refer to Figure 89 on page 74). Be sure to use precision resistors (0.1% or 1%) for differential signals, so that the restored DC levels on the positive and negative signals are very closely matched. In Figure 89 on page 74, the DC restore circuitry restores the DC level to 3.3 * 78.7/(140 + 78.7) = 1.1875 volts.

Figure 89. AC Coupling



Transceiver devices have DC biasing on the high-speed transceivers inputs and reference clock inputs designed for the 1.5-V PCML standard, so AC coupling is not required. This saves components and board space. If you are using other I/O

standards such as LVPECL or LVDS, then you need to AC-couple them, because their common mode voltage is different from the 1.5-V PCML common mode voltage. External biasing networks are not needed, because the common mode is generated internally in the device.

5.8. Additional Resources

- *Stratix III Signal Integrity white paper* (PDN section) (PDF)
- Signal & Power Integrity Design Techniques for SSN webcast

6. Memory Interfacing Guidelines

6.1. DDR3 Board Design Guidelines

- *AN 436: Design Guidelines for Implementing DDR3 SDRAM Interfaces in Stratix III Devices* (PDF)
- *AN 520: DDR3 SDRAM Memory Interface Termination and Layout Guidelines* (PDF)

6.2. DDR2 Board Design Guidelines

- *AN 328: Interfacing DDR2 SDRAM with Stratix II, Stratix II GX, and Arria GX Devices* (PDF)
- *AN 408: DDR2 Memory Interface Termination, Drive Strength, Loading, and Design Layout Guidelines* (PDF)
- *AN 444: Dual DIMM DDR2 SDRAM Memory Interface Design Guidelines* (PDF)
- *AN 445: Design Guidelines for Implementing DDR and DDR2 SDRAM Interfaces in Cyclone III Devices* (PDF)
- *AN 361: Interfacing DDR & DDR2 SDRAM with Cyclone II Devices* (PDF)

6.3. DDR Board Design Guidelines

- *AN 445: Design Guidelines for Implementing DDR and DDR2 SDRAM Interfaces in Cyclone III Devices* (PDF)
- *AN 361: Interfacing DDR & DDR2 SDRAM with Cyclone II Devices* (PDF)
- *AN 348: Interfacing DDR SDRAM with Cyclone Devices* (PDF)
- *AN 342: Interfacing DDR SDRAM with Stratix and Stratix GX Devices* (PDF)
- *AN 336: Using External Series and Parallel Termination with Stratix and Stratix GX Devices* (PDF)
- *AN 327: Interfacing DDR SDRAM with Stratix II Devices* (PDF)

6.4. QDR II and QDR II+

- *AN 326: Interfacing QDRII+ & QDRII with Stratix II Stratix II GX, Stratix, & Stratix GX Devices* (PDF)

6.5. RLDRAM II

- *AN 325: Interfacing RLDRAM II with Stratix II, Stratix, & Stratix GX Devices* (PDF)

6.6. Additional Resources

- Implementing High-Speed DDR3 Interfaces Webcast
- External Memory Solutions Center

7. Power Dissipation and Thermal Management

7.1. Power Management Guidelines

- Power Management Resource Center
- *AN 448: Stratix III Power Management Design Guide* (PDF)

7.2. Heat Sinking and Thermal Managment for FPGAs

- Thermal Resistance
- *AN 358: Thermal Management for FPGAs* (PDF)
- *AN 185: Thermal Management Using Heat Sinks* (PDF)

7.3. Additional Resources

- *High-Speed Board Design Advisor: Thermal Management* technical brief (PDF)
  Stratix III Programmable Power white paper (PDF)

8. Tools, Models, and Libraries

8.1. Design Tools

- Early Power Estimator
- PDN Design Tool
- PDN Design Tool User Guide
- Stratix V Early SSN Estimator and user guide (PDF)
- Stratix II GX Early SSN Estimator and user guide (PDF)
- Stratix III Early SSN Estimator and user guide (PDF)
- Arria V Early SSN Estimator and user guide (PDF)

8.2. I/O Buffer Models

- IBIS
- HSPICE

8.3. Device Libraries

- Orcad Symbols by Device and Package

8.4. Net Length Reports

- Intel Agilex Net Length Information
- Intel Stratix 10 Net Length Information
- Intel MAX 10 Net Length Information
- Stratix V Net Length Information
- Stratix IV GT Net Length Information
- Stratix IV GX Net Length Information
- Stratix IV E Net Length Information
- Stratix III Net Length Information
- Stratix II GX Net Length Information
- Stratix II Net Length Information
- HardCopy® II Net Length Information
- Intel Cyclone 10 GX Net Length Information
- Intel Cyclone 10 LP Net Length Information

- Cyclone V Net Length Information
- Cyclone IV SoC Net Length Information
- Cyclone III Net Length Information
- Cyclone II Net Length Information
- Intel Arria 10 Net Length Information
- Arria V GX/GT Net Length Information
- Arria V GZ Net Length Information
- Arria V SoC Net Length Information
- Arria II GX Net Length Information
- Arria GX Net Length Information

8.5. Thermal Models

- Stratix IV GT Flowtherm Models
- Stratix IV GX Flowtherm Models
- Stratix IV E Flowtherm Models
- Stratix III Flotherm Models
- Stratix II Flotherm Models
- Cyclone III Flotherm Models

9. Reference Designs and Development Kits

9.1. Development Kits

- Intel® Stratix® 10 TX Signal Integrity Development Kit
- Intel Stratix 10 SX SoC Development Kit
- Intel Stratix 10 MX FPGA Development Kit
- Intel Stratix 10 TX Transceiver Signal Integrity Development Kit
- Intel Stratix 10 GX FPGA Development Kit
- Intel® Agilex™ Transceiver-SoC Development Kit
- Intel Agilex F-Series FPGA Development Kit
- Arria® II GX FPGA Development Kit
- Audio Video Development Kit, Stratix IV GX Edition
- Cyclone® III LS FPGA Development Kit
- DSP Development Kit, Cyclone III Edition
- DSP Development Kit, Stratix III Edition
- Embedded Systems Development Kit, Cyclone III Edition
- Nios® II Embedded Evaluation Kit, Cyclone III Edition
- Stratix IV E FPGA Development Kit
- Stratix IV GX FPGA Development Kit
- Transceiver Signal Integrity Development Kit, Stratix IV GX Edition
- PCI Express Development Kit, Stratix II GX Edition
- PCI Development Kit, Cyclone II Edition
- MAX® II Development Kit
- Cyclone III FPGA Starter Kit
- Cyclone III FPGA Development Kit
- Stratix III FPGA Development Kit

9.2. Development Kits by Technology

9.2.1. DDR3 Memory Technology

- Stratix IV GX FPGA Development kit
- Stratix IV E FPGA Development kit
- Arria II GX FPGA Development kit
- Audio Video Stratix IV GX Edition Development kit

9.2.2. DDR2 Memory Technology

- Stratix III FPGA Development Kit
- Arria II GX FPGA Development Kit
- Cyclone III LS FPGA Development Kit
- Cyclone III FPGA Development Kit
- DSP Development Kit, Stratix III Edition
- DSP Development Kit, Cyclone III Edition
- Embedded Systems Development Kit, Cyclone III Edition
- PCI Development Kit, Cyclone II Edition

9.2.3. DDR Memory Technology

- Cyclone III FPGA Starter Kit
- Nios II Embedded Evaluation kit, Cyclone III Edition

9.2.4. QDR2 SRAM Memory Technology

- Stratix IV GX FPGA Development Kit
- Stratix IV E FPGA Development Kit
- Audio Video Stratix IV GX Edition Development Kit
- DSP Development Kit, Stratix III Edition
- Stratix III FPGA Development Kit

9.2.5. SRAM Memory Technology

- Stratix IV GX FPGA Development Kit
- Stratix IV E FPGA Development Kit
- Audio Video Stratix IV GX Edition Development Kit
- Arria II GX FPGA Development Kit
- Cyclone III FPGA Development Kit
- Cyclone III LS FPGA Development Kit
- Cyclone III FPGA Starter Kit

- DSP Development Kit, Cyclone III Edition
- MAX II Development Kit
- Nios II Embedded Evaluation Kit, Cyclone III Edition

9.2.6. RLDRAM II Memory Technology

- Stratix IV E FPGA Development Kit

9.2.7. FLASH Memory Technology

- Stratix IV GX FPGA Development Kit
- Stratix IV E FPGA Development Kit
- Arria II GX FPGA Development Kit
- Audio Video Stratix IV GX Edition Development Kit
- Cyclone III FPGA Development Kit
- Cyclone III LS FPGA Development Kit
- Cyclone III FPGA Starter Kit
- DSP Development Kit, Cyclone III Edition
- DSP Development Kit, Stratix III Edition
- Embedded Systems Development Kit, Cyclone III Edition
- Nios II Embedded Evaluation Kit, Cyclone III Edition
- Stratix III FPGA Development Kit
- Transceiver Signal Integrity Development Kit, Stratix IV GX Edition

9.2.8. Gigabit Transceiver Technology

- Transceiver Signal Integrity Development Kit, Stratix IV GX Edition

9.2.9. PCI Express Technology

- Stratix IV GX FPGA Development Kit

9.2.10. PCI Technology

- MAX II Development Kit
- PCI Development Kit, Cyclone II Edition

9.2.11. 10/100/1000 Ethernet Technology

- Stratix IV GX FPGA Development Kit
- Stratix IV E FPGA Development Kit
- Cyclone III FPGA Development Kit
- Cyclone III LS FPGA Development Kit
- DSP Development Kit, Cyclone III Edition

- Embedded Systems Development Kit, Cyclone III Edition
- Stratix III FPGA Development Kit
- Transceiver Signal Integrity Development Kit, Stratix IV GX Edition

9.2.12. USB 2.0 Technology

- Cyclone III FPGA Development Kit
- DSP Development Kit, Cyclone III Edition
- DSP Development Kit, Stratix III Edition
- Embedded Systems Development Kit, Cyclone III Edition
- Stratix III FPGA Development Kit

10. Document Revision History for AN 958: Board Design Guidelines

Document Version	Changes
2022.01.28	Updated for latest Intel branding standards.
2021.09.27	Initial release.