Application Report
SPRAAR7D–August 2014–Revised March 2015

USB Layout Guidelines

Embedded Processor Applications

ABSTRACT

The universal serial bus (USB) is an interface for connecting peripheral devices to host computers. As of version 3.1 of the USB specification, SuperSpeed+ mode supports speeds of up to 10 Gbps through a differential data pair that operates at frequencies of up to 5 GHz. Because of these high-frequency signals, take care to a maintain signal quality when committing a schematic design to printed-circuit-board (PCB).

Contents

List of Figures

List of Tables

1 Introduction

1.1 Scope

This application report can help system designers implement best practices and understand PCB layout options when designing platforms using the USB interface. This document is intended for audiences familiar with PCB manufacturing, layout, and design.

1.2 Critical Signals

A primary concern when designing a system is accommodating and isolating high-speed signals. As high-speed signals are most likely to impact or be impacted by other signals, they must be laid out early (preferably first) in the PCB design process to ensure that prescribed routing rules can be followed.

Table 1 outlines the signals requiring the most attention in a USB layout.

Table 1. Critical Signals

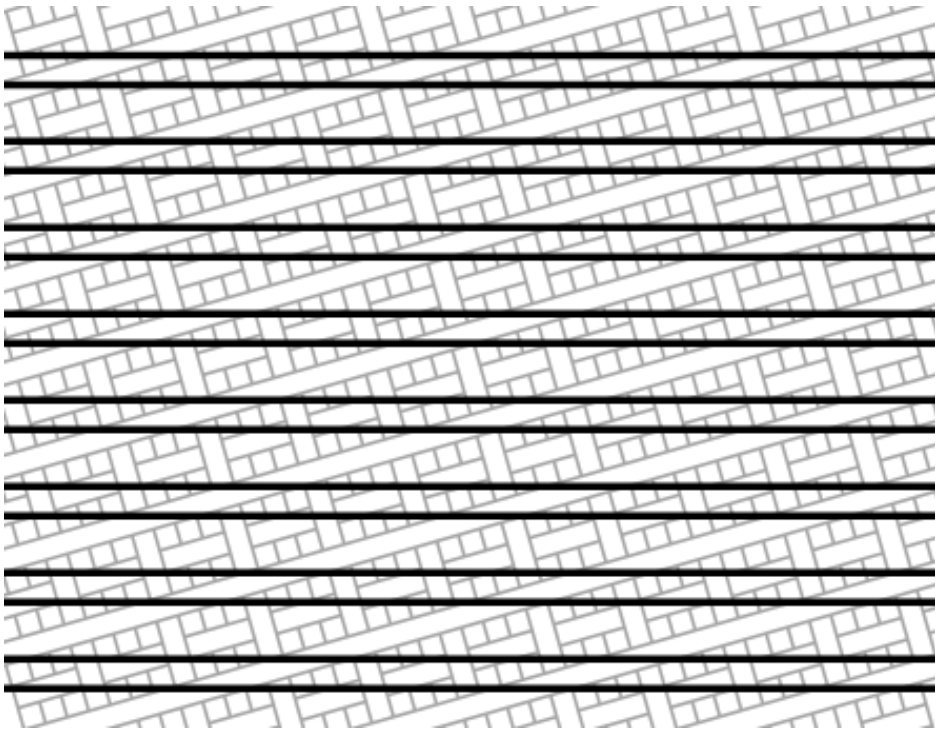
Signal Name	Description
DP	USB 2.0 differential pair, positive
DM	USB 2.0 differential pair, negative
SSTXP	SuperSpeed differential pair, TX, positive
SSTXN	SuperSpeed differential pair, TX, negative
SSRXP	SuperSpeed differential pair, RX, positive
SSRXN	SuperSpeed differential pair, RX, negative

2 General High-Speed Signal Routing

2.1 PCB Fiber Weave Mitigation

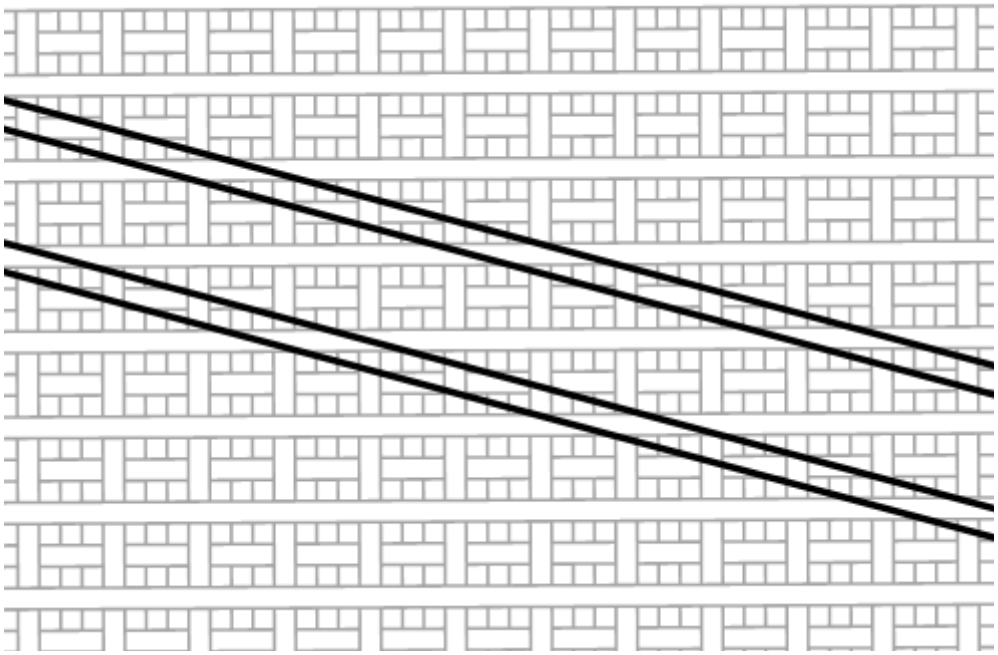
When routing differential signals across common PCB materials, each trace of the pair will experience different dielectric constants and corresponding signal velocities due to the differences in static permittivity (Ɛr) of the fiberglass weave (Ɛr is approximately 6) and epoxy (Ɛr is approximately 3) that comprise a PCB. As signals travel faster when Ɛr is lower, an interpair skew can develop if a signal in a differential pair travels over a higher ratio of fiberglass or epoxy than does its companion signal. This skew between the differential signals can significantly degrade the differential eye diagram as presented to the receiver, cause significant AC common-mode voltage noise, and cause EMI issues. The extent of this problem will depend on the bus speed, the length of the traces, the trace geometries, the type of fiberglass weave used, and the alignment of the traces to the weave pattern of a PCB. Problems from fiber weave alignment vary from board to board. This variance makes issues difficult to diagnose.

Figure 1, Figure 2, and Figure 3 show the three most common methods to minimize the impact of PCB fiber weave in a board design. The goal of each method is to ensure that both signals of the differential pair will share a relatively common Ɛr across the length of the pair routing.



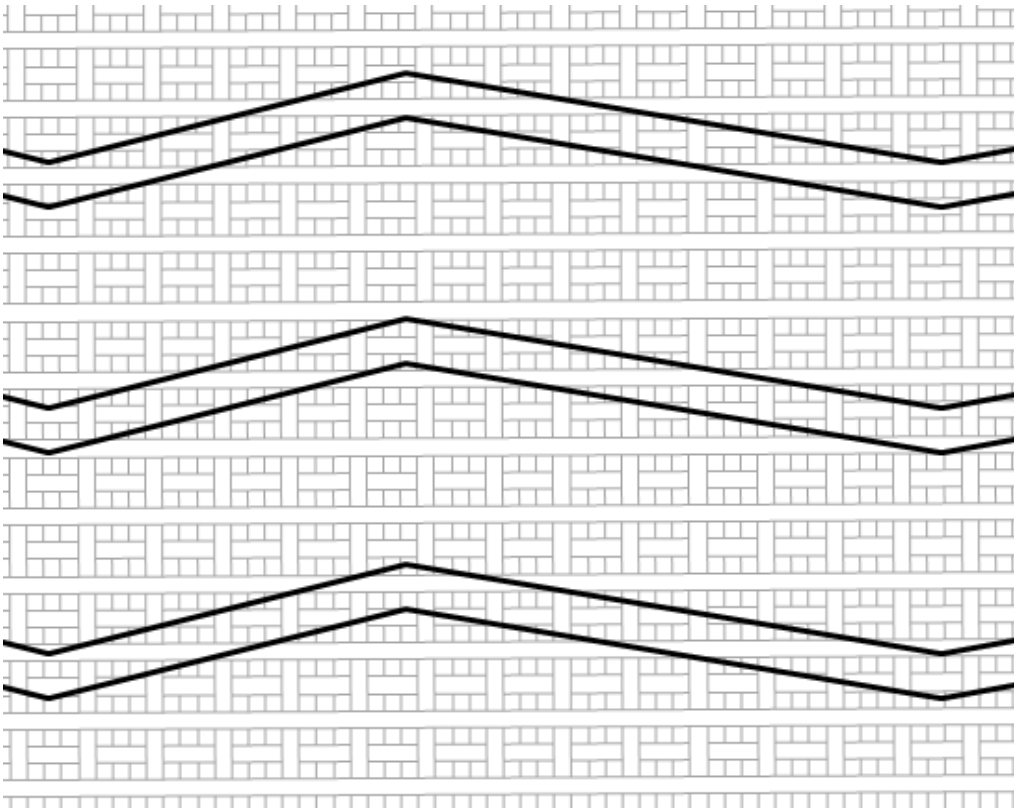
The entirety of the signaling image plane is rotated 10° to 35° in relation to the underlying PCB fiber weave. The PCB manufacturer can affect this rotation without making changes to the PCB layout database.

Figure 1. Rotation of the PCB Image



Only the USB differential signals are routed at a 10° to 35° angle in relation to the underlying PCB fiber weave.

Figure 2. Routing Angle Rotation



The differential signals are routed in a zig-zag fashion across the PCB.

Figure 3. Zig-Zag Routing

Because the ratio of fiberglass to epoxy is the primary contributor to the Ɛr disparity, choose a PCB style with a tighter weave, less epoxy, and greater Ɛr uniformity across longer trace lengths. Before sending your design out for fabrication, specify a PCB style that can best accommodate high-speed signals. For examples of common PCB styles, see Figure 4.

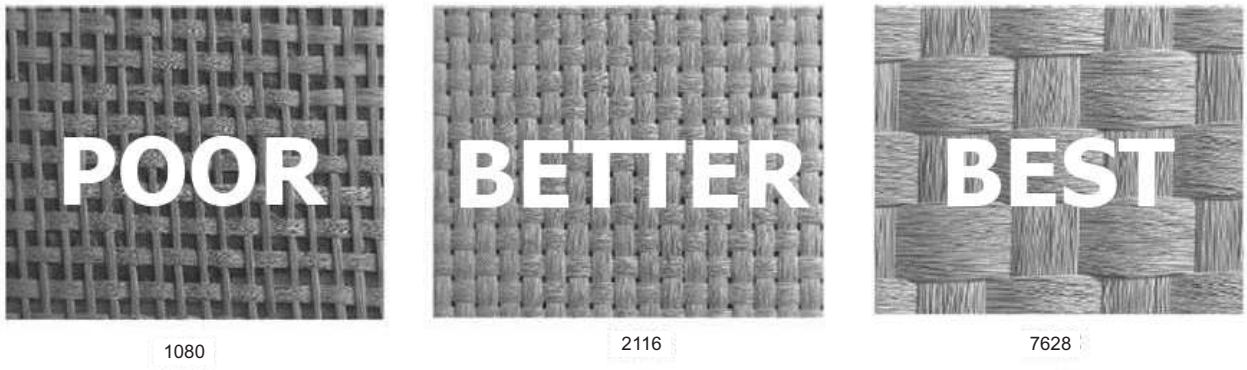


Figure 4. PCB Fiberglass Style Examples

2.2 USB Signal Trace Lengths

As with all high-speed signals, keep total trace length for signal pairs to a minimum. For trace length requirements for each device, see Appendix A.

2.3 USB Signal Trace Length Matching

Match the etch lengths of the differential pair traces (that is, DP and DM, SSRXP and SSRXN, or SSTXP and SSTXN). The etch length of the differential pair groups do not need to match (that is, the length of the SSRX pair does not need to match the length of the SSTX pair). When matching the intrapair length of the USB signals, add serpentine routing to match the lengths as close to the mismatched ends as possible. See Figure 5 for more details.

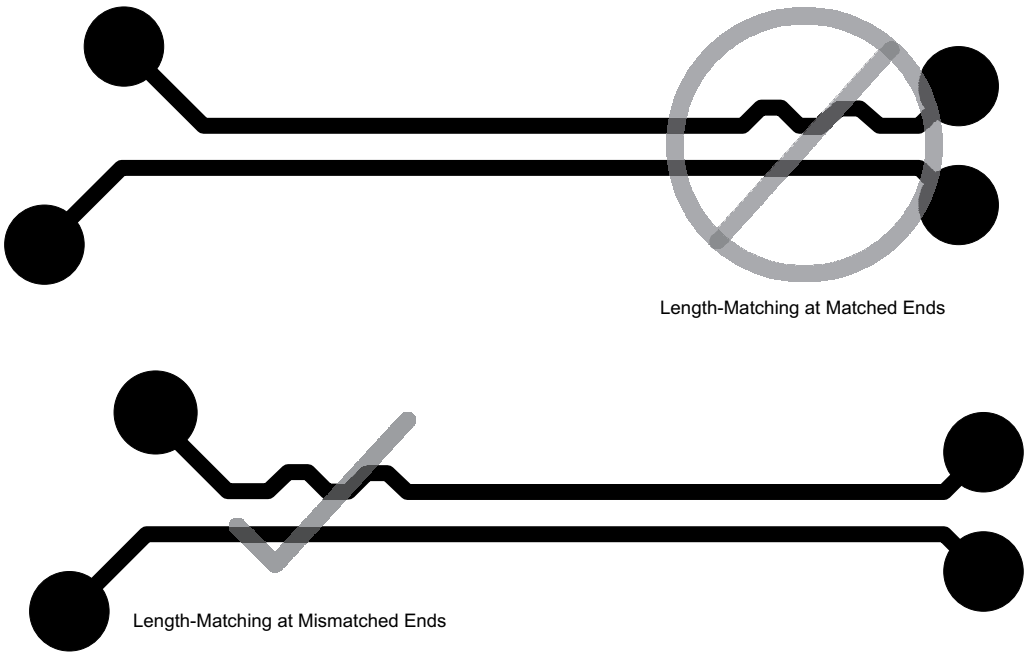


Figure 5. Length Matching

2.4 USB Signal Reference Planes

USB signals should be routed over a solid GND reference plane and not across a plane split or a void in the reference plane unless absolutely necessary. TI does not recommend USB signal references to power planes.

Routing across a plane split or a void in the reference plane forces return high-frequency current to flow around the split or void. This can result in the following conditions:

- Excess radiated emissions from an unbalanced current flow
- Delays in signal propagation delays due to increased series inductance
- Interference with adjacent signals
- Degraded signal integrity (that is, more jitter and reduced signal amplitude)

For examples of correct and incorrect plane void routing, see Figure 6 and Figure 7.

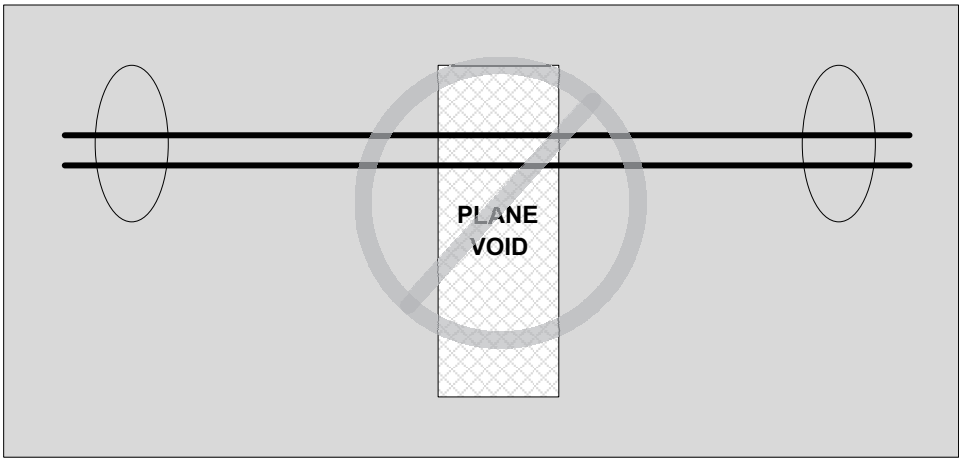


Figure 6. Incorrect Plane Void Routing

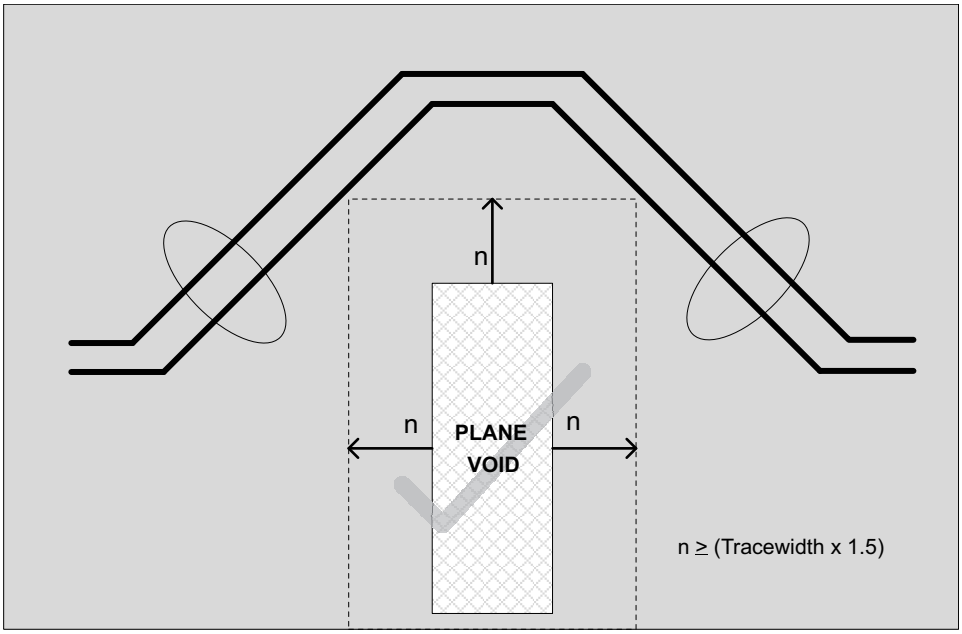


Figure 7. Correct Plane Void Routing

If routing over a plane-split is completely unavoidable, place stitching capacitors across the split to provide a return path for the high-frequency current. These stitching capacitors minimize the current loop area and any impedance discontinuity created by crossing the split. These capacitors should be 1 µF or lower and placed as close as possible to the plane crossing. For examples of incorrect plane-split routing and correct stitch capacitor placement, see Figure 8 and Figure 9.

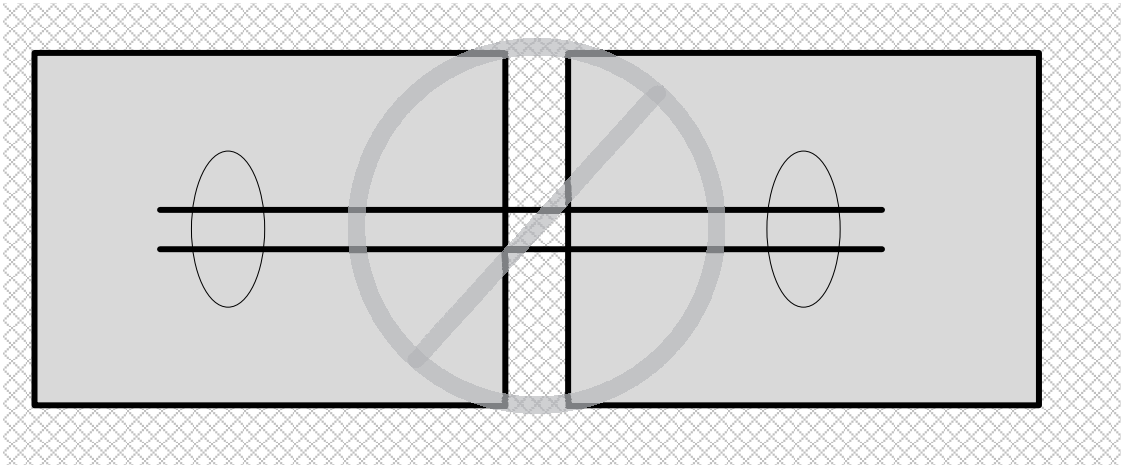


Figure 8. Incorrect Plane-Split Signal Routing

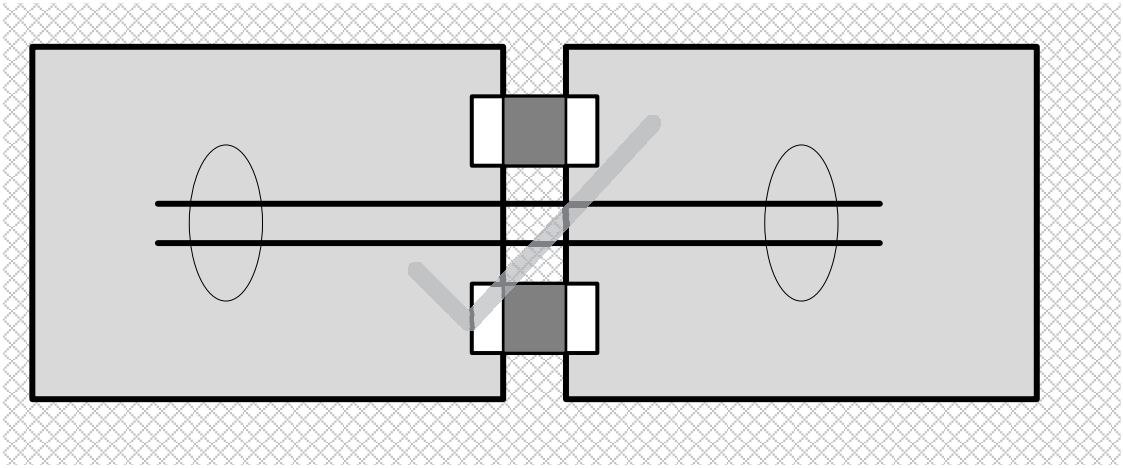


Figure 9. Stitching Capacitor Placement

When planning a PCB stackup, ensure that planes that do not reference each other are not overlapped because this produces unwanted capacitance between the overlapping areas. To see an example of how this capacitance could pass RF emissions from one plane to the other, see Figure 10.

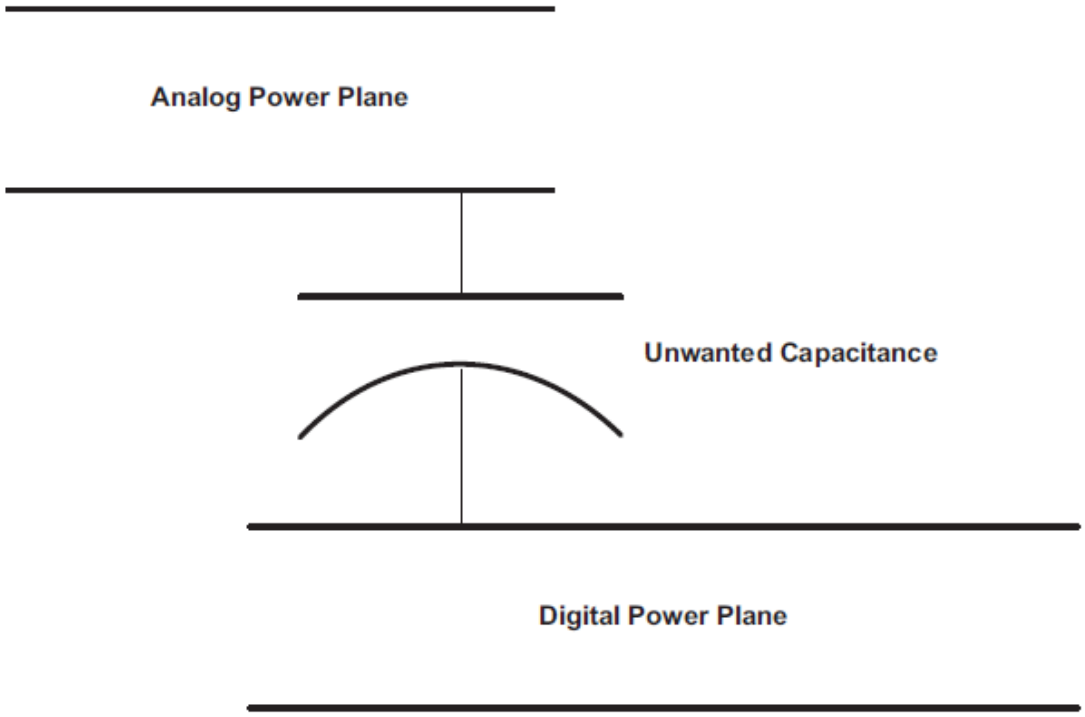


Figure 10. Overlapped Planes

The entirety of any USB signal trace should maintain the same GND reference from origination to termination. If unable to maintain the same GND reference, via-stitch both GND planes together to ensure continuous grounding and uniform impedance. Place these stitching vias symmetrically within 200 mils (center-to-center, closer is better) of the signal transition vias. For an example of stitching vias, see Figure 11.

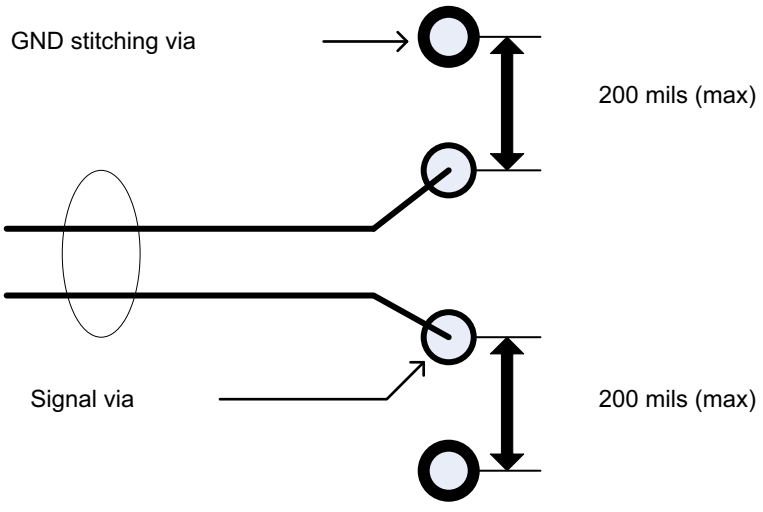


Figure 11. Stitching Vias

3 USB Differential Signal Routing

3.1 Differential Signal Spacing

To minimize crosstalk in USB implementations, the spacing between the signal pairs must be a minimum of 5 times the width of the trace. This spacing is the 5W rule. A PCB design with a calculated trace width of 6 mils requires a minimum of 30 mils spacing between the SuperSpeed pairs. Also, maintain a minimum keep-out area of 30 mils to any other signal throughout the length of the trace. Where the USB differential pair abuts a clock or a periodic signal, increase this keep-out to a minimum of 50 mils to ensure proper isolation. For examples of USB3 and USB2 differential signal spacing, see Figure 12 and Figure 13.

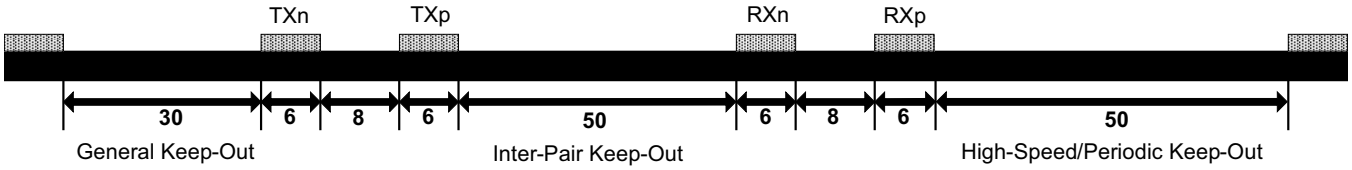


Figure 12. USB3 Differential Signal Spacing (mils)

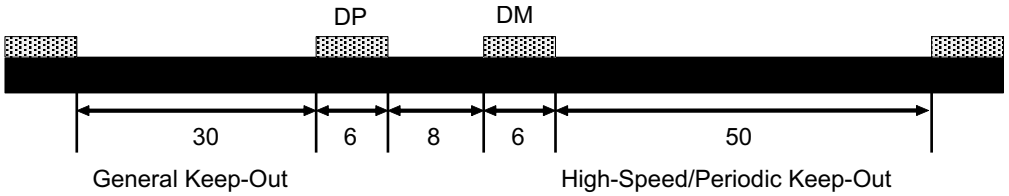


Figure 13. USB2 Differential Signal Spacing (mils)

3.2 Differential Signal Rules

- Do not place probe or test points on any USB differential signal.
- Do not route USB traces under or near crystals, oscillators, clock signal generators, switching power regulators, mounting holes, magnetic devices, or ICs that use or duplicate clock signals.
- After BGA breakout, keep USB differential signals clear of the SoC because high current transients produced during internal state transitions can be difficult to filter out.
- When possible, route the USB differential pair signals on the top or bottom layer of the PCB with an adjacent GND layer. TI does not recommend stripline routing of the USB differential signals.
- Ensure that USB differential signals are routed ≥ 90 mils from the edge of the reference plane.
- Ensure that USB differential signals are routed at least 1.5 W (calculated trace-width × 1.5) away from voids in the reference plane. This rule does not apply where SMD pads on the USB differential signals are voided.
- Maintain constant trace width after the SoC BGA escape to avoid impedance mismatches in the transmission lines.
- Maximize differential pair-to-pair spacing when possible.

3.3 Symmetry in the Differential Pairs

Route all USB differential pairs together symmetrically and parallel to each other. Deviating from this requirement will occur naturally during package escape and when routing to connector pins. These deviations should be as short as possible and package break-out should occur within 0.25 in. of the package.

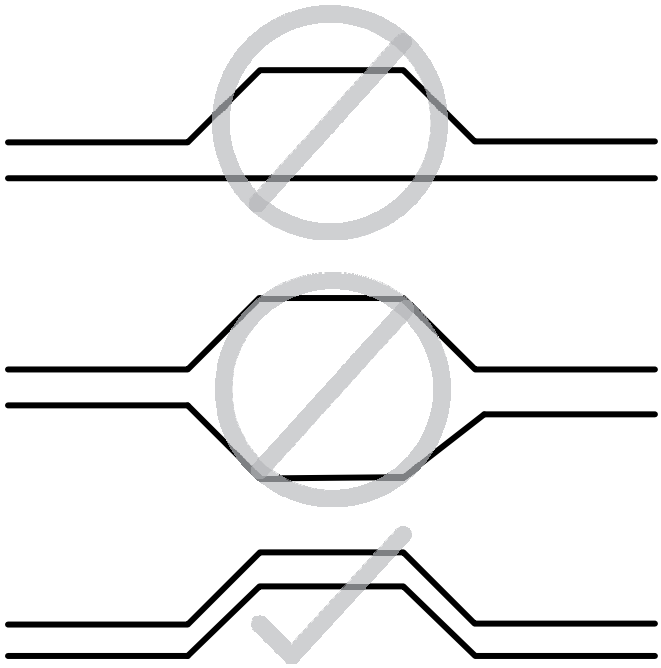


Figure 14. Differential Pair Symmetry

3.4 Crosstalk Between the Differential Signal Pairs

In devices that include both USB 2.0 and USB 3.0 interfaces, avoiding crosstalk between interfaces is important. To avoid crosstalk, ensure that the USB2.0 DP/DM pair and the USB3.0 SSTX/SSRX pairs are not routed within 50 mils of each other after package escape and before connector termination.

3.5 USB Receptacle

When implementing a USB through-hole receptacle (like a USB Standard-A), TI recommends making USB differential signal connections to the receptacle on the bottom layer of the PCB. Making these connections on the bottom layer of the PCB prevents the through-hole pin from acting as a stub in the transmission path. For surface-mount receptacles such as USB Micro-B and Micro-AB, make USB differential signal connections on the top layer. Making these connections on the top layer eliminates the need for vias in the transmission path. For examples of USB through-hole receptacle connections, see Figure 15.

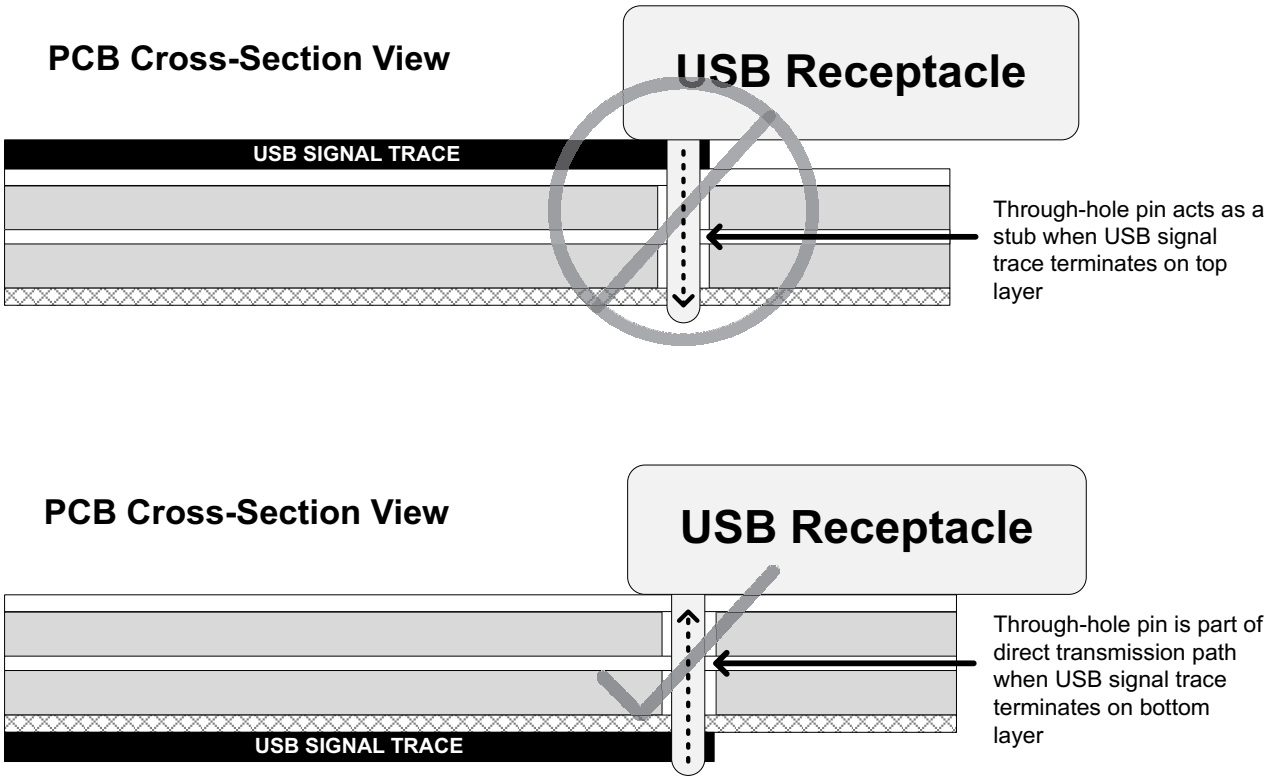


Figure 15. USB Through-Hole Receptacle Connection

3.6 Via Discontinuity Mitigation

A via presents a short section of change in geometry to a trace and can appear as a capacitive and/or an inductive discontinuity. These discontinuities result in reflections and some degradation of a signal as it travels through the via. Reduce the overall via stub length to minimize the negative impacts of vias (and associated via stubs).

Because longer via stubs resonate at lower frequencies and increase insertion loss, keep these stubs as short as possible. In most cases, the stub portion of the via present significantly more signal degradation than the signal portion of the via. TI recommends keeping via stubs to less than 15 mils. Longer stubs must be back-drilled. For examples of short and long via lengths, see Figure 16 and Figure 17.

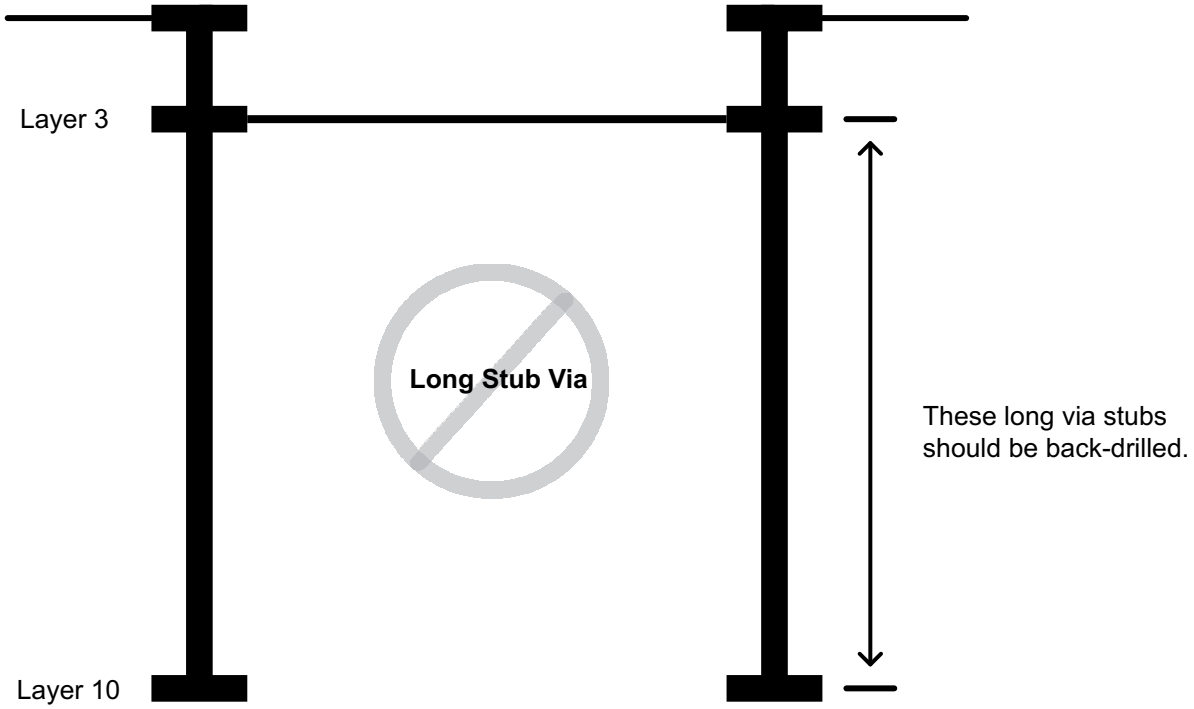


Figure 16. Via Length (Long Stub)

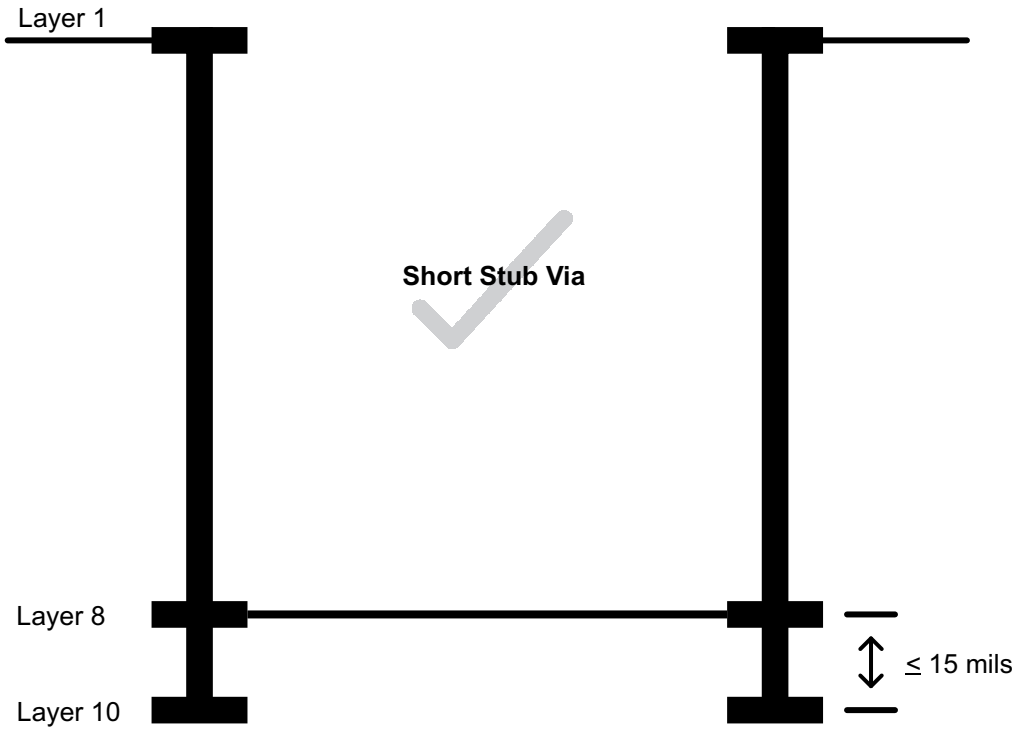


Figure 17. Via Length (Short Stub)

3.7 Back-Drill Stubs

Back-drilling is a PCB manufacturing process in which the undesired conductive plating in the stub section of a via is removed. To Back-drill, use a drill bit slightly larger in diameter than the drill bit used to create the original via hole. When via transitions result in stubs longer than 15 mils, back-drill the resulting stubs to reduce insertion losses and to ensure that they do not resonate.

3.8 Increase Via Anti-Pad Diameter

Increasing the via anti-pad diameter reduces the capacitive effects of the via and the overall insertion loss. Ensure that anti-pad diameter for vias on any high-speed signal are as large as possible (50 mils provides significant benefits without imposing undue implementation hardship). The copper clearance, indicated by this anti-pad, must be met on all layers where the via exists, including both routing layer and plane layers. The traces connecting to the via barrel contain the only copper allowed in this area. For an example of a via anti-pad diameter, see Figure 18.

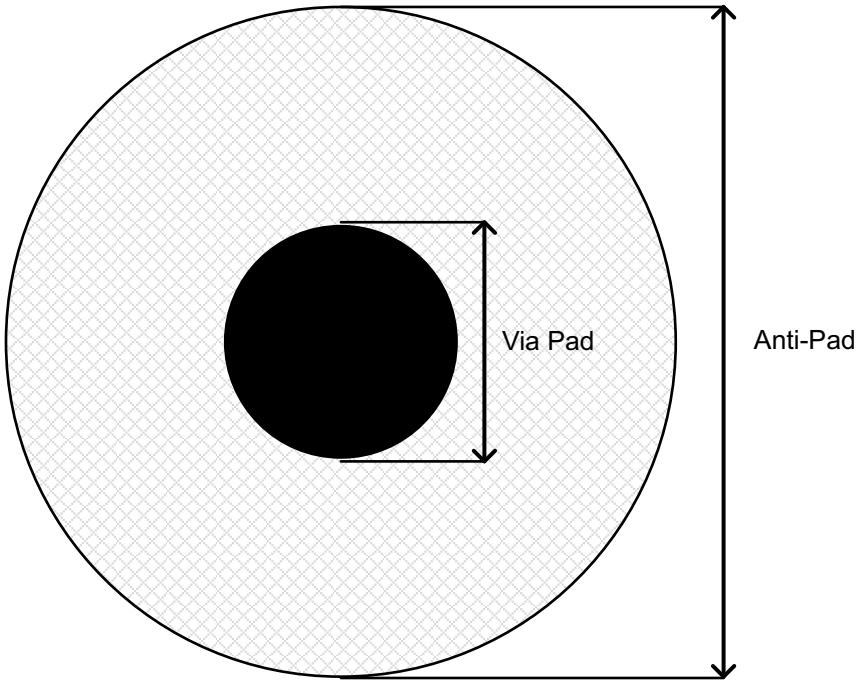


Figure 18. Anti-Pad Diameter

3.9 Equalize Via Count

If using vias is necessary on a USB differential signal trace, ensure that the via count on each member of the differential pair is equal and that the vias are as evenly spaced as possible. TI recommends placing vias as close as possible to the SoC.

3.10 Surface-Mount Device Pad Discontinuity Mitigation

Avoid including surface-mount devices (SMDs) on high-speed signal traces because these devices introduce discontinuities that can negatively affect signal quality. When SMDs are required on the signal traces (for example, the USB SuperSpeed transmit AC coupling capacitors) the maximum permitted component size is 0603. TI strongly recommends using 0402 or smaller. Place these components symmetrically during the layout process to ensure optimum signal quality and to minimize reflection. For examples of correct and incorrect AC coupling capacitor placement, see Figure 19.

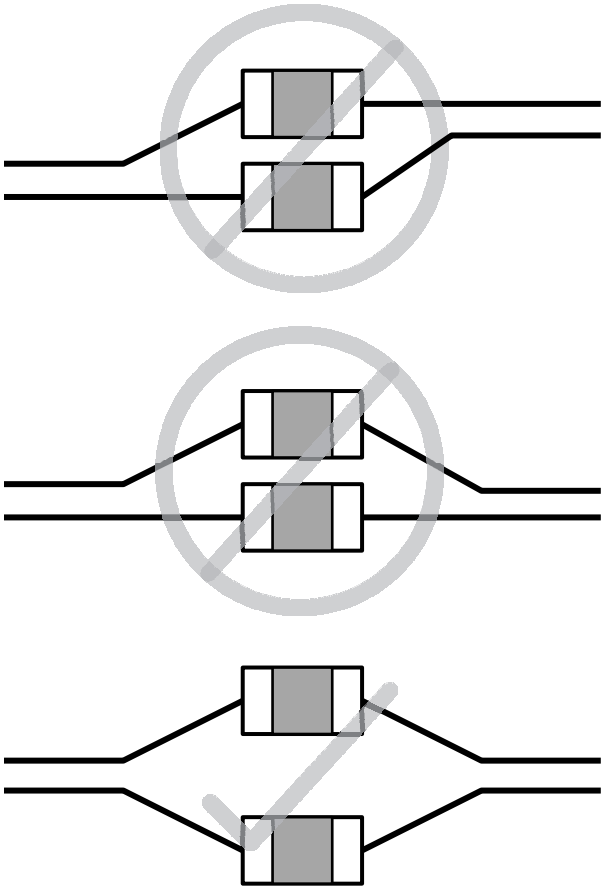


Figure 19. AC-Coupling Placement

To minimize the discontinuities associated with the placement of these components on the differential signal traces, TI recommends partially voiding these SMD mounting pads of the reference plane by approximately 60% because this value strikes a balance between the capacitive effects of a 0% reference void and the inductive effects of a 100% reference void. This void should be at least two PCB layers deep. For an example of a reference plane voiding of surface mount devices, see Figure 20.

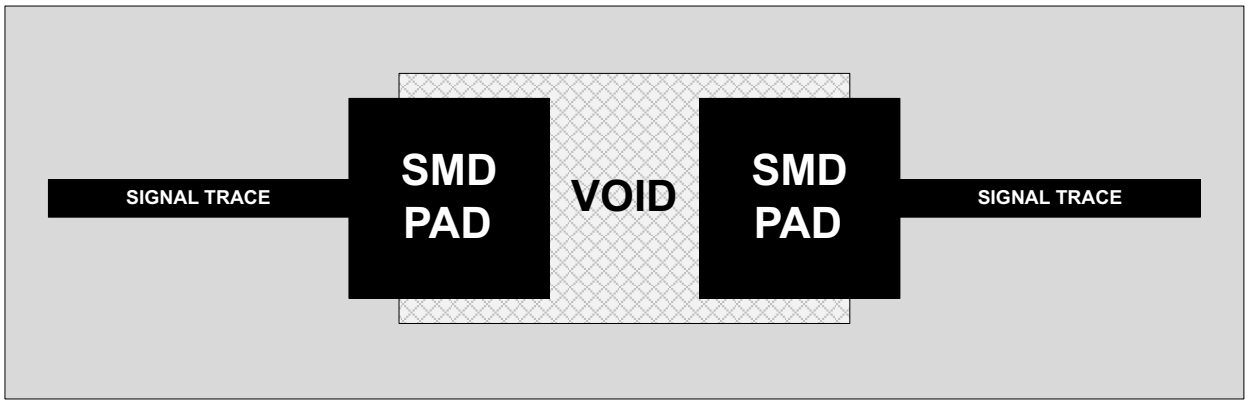


Figure 20. Reference Plane Voiding of Surface-Mount Devices

3.11 Signal Bending

Avoid the introduction of bends into the USB differential pairs. When bending is required, maintain a bend angle greater than 135° to ensure that the bend is as loose as a possible. For an example of USB signal bending rules, see Figure 21.

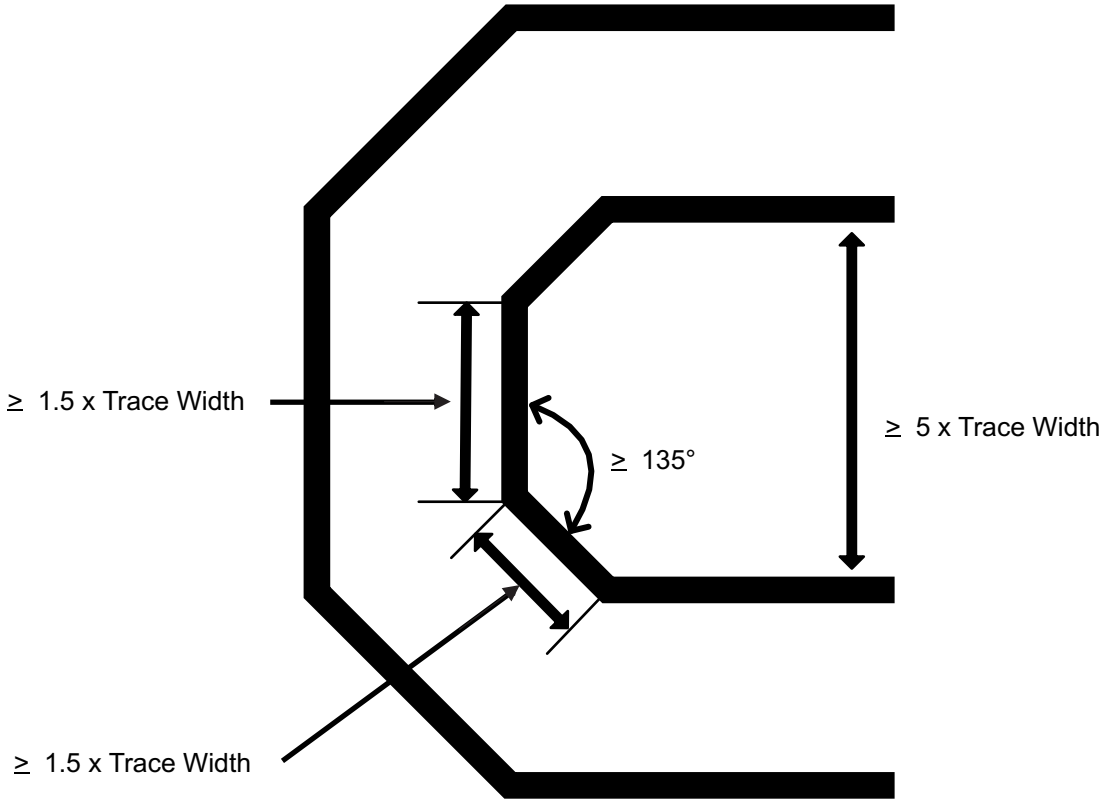


Figure 21. Signal Bending Rules

3.12 Suggested PCB Stackups

Due to the high frequencies associated with SuperSpeed USB, TI recommends a PCB of at least six layers. Table 2 provides example PCB stackups.

Table 2. Example PCB Stackups

6-LAYER	8-LAYER	10-LAYER
SIGNAL	SIGNAL	SIGNAL
GROUND	GROUND	GROUND
SIGNAL(1)	SIGNAL	SIGNAL(1)
SIGNAL(1)	SIGNAL	SIGNAL(1)
POWER/GROUND(2)	POWER/GROUND(2)	POWER
SIGNAL	SIGNAL	POWER/GROUND(2)
	GROUND	SIGNAL(1)
	SIGNAL	SIGNAL(1)
		GROUND
		SIGNAL

(1) Route directly adjacent signal layers at a 90° offset to each other

(2) Plane may be split depending on specific board considerations. Ensure that traces on adjacent planes do not cross splits.

3.13 ESD/EMI Considerations

When choosing ESD/EMI components, TI recommends selecting devices that permit flow-through routing of the USB differential signal pair because they provide the cleanest routing. For example, the TI TPD4EUSB30 can be combined with the TI TPD2EUSB30 to provide flow-through ESD protection for both USB2 and USB3 differential signals without the need for bends in the signal pairs. For an example of flow-through routing, see Figure 22.

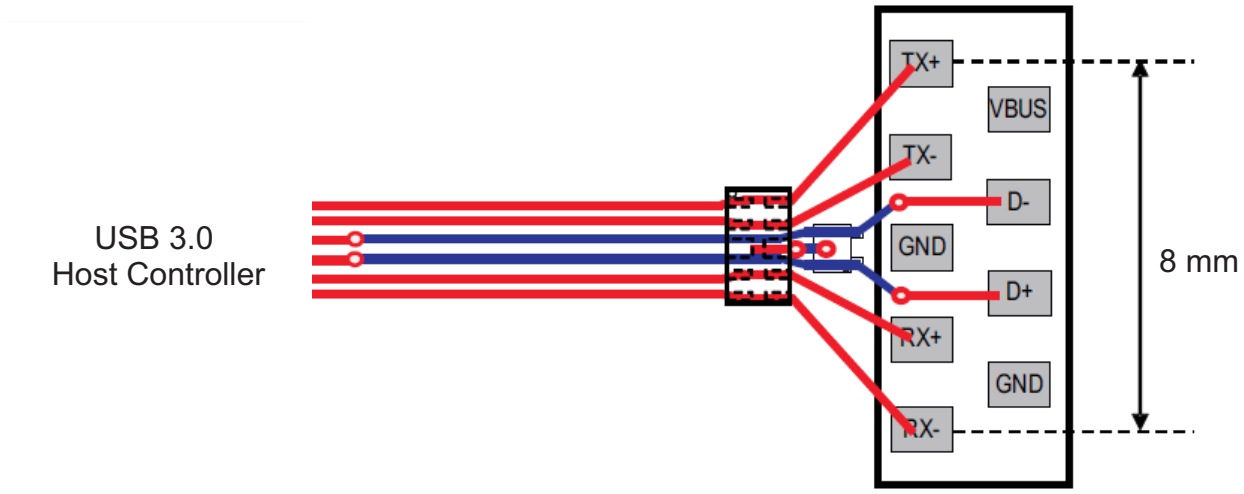


Figure 22. Flow-Through Routing

3.14 ESD/EMI Layout Rules

- Place ESD and EMI protection devices as close as possible to the connector.
- Keep any unprotected traces away from protected traces to minimize EMI coupling.
- Incorporate 60% voids under the ESD/EMI component signal pads to reduce losses.
- Use 0402 0-Ω resistors for common-mode filter (CMF) no-stuff options because larger components will typically introduce more loss that the CMF itself.
- Place the SuperSpeed transmit pair AC coupling capacitors on the protected side of the CMF and as close as possible to the CMF.
- If vias are needed to transition to the CMF layer, ensure that the vias are as close as possible to the CMF.
- Keep the overall routing of AC-coupling capacitors + CMF + ESD protection as short and as close as possible to the connector.

Device Layout Parameters

Table 3. AM335x/AM437x

Parameter	MIN	TYP	MAX	Unit
SuperSpeed Tracelength (total)	NA			
USB2.0 Tracelength (total)		4000	12000	Mils
Skew within any SuperSpeed differential pair	NA			
Skew within any USB2.0 differential pair			50	Mils
USB2.0 DP/DM pair differential impedance	81	90	99	Ω
USB2.0 DP/DM pair common-mode impedance	40.5	45	49.5	Ω
SuperSpeed SSRX/SSTX pair differential impedance	NA			
Number of stubs allowed on any USB differential pair trace (total)			0	Stubs
Number of vias allowed on each SuperSpeed differential trace (total)	NA			
Number of vias allowed on each USB2.0 differential trace (total)			4	Vias
Number of test points permitted on any USB differential pair trace (total)			0	Test Points
USB differential pair to clock or high-speed periodic signal trace spacing	50			Mils
USB differential pair to any other signal trace spacing	30			Mils

Table 4. DRA7xx/AM57xx

Parameter	MIN	TYP	MAX	Unit
SuperSpeed Tracelength (total)			3500	Mils
USB2.0 Tracelength (total)		4000	12000	Mils
Skew within any SuperSpeed differential pair			5	Mils
Skew within any USB2.0 differential pair			50	Mils
USB2.0 DP/DM pair differential impedance	81	90	99	Ω
USB2.0 DP/DM pair common-mode impedance	40.5	45	49.5	Ω
SuperSpeed SSRX/SSTX pair differential impedance	83.7	90	96.3	Ω
Number of stubs allowed on any USB differential pair trace (total)			0	Stubs
Number of vias allowed on each SuperSpeed differential trace (total)			2	Vias
Number of vias allowed on each USB2.0 differential trace (total)			4	Vias
Number of test points permitted on any USB differential pair trace (total)			0	Test Points
USB differential pair to clock or high-speed periodic signal trace spacing	50			Mils
USB differential pair to any other signal trace spacing	30			Mils

Table 5. KeyStone II - K2K, K2H, K2L and K2E Devices

Parameter	MIN	TYP	MAX	UNIT
SuperSpeed Tracelength (Total, nominal PCB material)			5500	Mils
SuperSpeed Insertion Loss at 2.5 GHz (device to connector)			10	dB
USB 2.0 Tracelength (Total)			12000	Mils
Skew within any SuperSpeed differential pair			5	Mils
Skew within any USB 2.0 differential pair			50	Mils
USB 2.0 DP/DM pair differential impedance	81	90	99	Ω
USB 2.0 DP/DM pair common mode impedance	40.5	45	49.5	Ω
SuperSpeed SSRX/SSTX pair differential impedance	83.7	90	96.3	Ω
Number of stubs allowed on any USB differential pair trace (Total)			0	Stubs
Number of vias allowed on each SuperSpeed differential trace (Total)			2	Vias
Number of vias allowed on each USB 2.0 differential trace (Total)			4	Vias
Number of test points permitted on any USB differential pair trace (Total)			0	Test Points
USB differential pair to clock or high-speed periodic signal trace spacing	50			Mils
USB differential pair to any other signal trace spacing	30			Mils

Revision History

Changes from C Revision (August 2014) to D Revision	Page
• Updated document with new content	4

IMPORTANT NOTICE

Texas Instruments Incorporated and its subsidiaries (TI) reserve the right to make corrections, enhancements, improvements and other changes to its semiconductor products and services per JESD46, latest issue, and to discontinue any product or service per JESD48, latest issue. Buyers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. All semiconductor products (also referred to herein as “components”) are sold subject to TI’s terms and conditions of sale supplied at the time of order acknowledgment.

TI warrants performance of its components to the specifications applicable at the time of sale, in accordance with the warranty in TI’s terms and conditions of sale of semiconductor products. Testing and other quality control techniques are used to the extent TI deems necessary to support this warranty. Except where mandated by applicable law, testing of all parameters of each component is not necessarily performed.

TI assumes no liability for applications assistance or the design of Buyers’ products. Buyers are responsible for their products and applications using TI components. To minimize the risks associated with Buyers’ products and applications, Buyers should provide adequate design and operating safeguards.

TI does not warrant or represent that any license, either express or implied, is granted under any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI components or services are used. Information published by TI regarding third-party products or services does not constitute a license to use such products or services or a warranty or endorsement thereof. Use of such information may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

Reproduction of significant portions of TI information in TI data books or data sheets is permissible only if reproduction is without alteration and is accompanied by all associated warranties, conditions, limitations, and notices. TI is not responsible or liable for such altered documentation. Information of third parties may be subject to additional restrictions.

Resale of TI components or services with statements different from or beyond the parameters stated by TI for that component or service voids all express and any implied warranties for the associated TI component or service and is an unfair and deceptive business practice. TI is not responsible or liable for any such statements.

Buyer acknowledges and agrees that it is solely responsible for compliance with all legal, regulatory and safety-related requirements concerning its products, and any use of TI components in its applications, notwithstanding any applications-related information or support that may be provided by TI. Buyer represents and agrees that it has all the necessary expertise to create and implement safeguards which anticipate dangerous consequences of failures, monitor failures and their consequences, lessen the likelihood of failures that might cause harm and take appropriate remedial actions. Buyer will fully indemnify TI and its representatives against any damages arising out of the use of any TI components in safety-critical applications.

In some cases, TI components may be promoted specifically to facilitate safety-related applications. With such components, TI’s goal is to help enable customers to design and create their own end-product solutions that meet applicable functional safety standards and requirements. Nonetheless, such components are subject to these terms.

No TI components are authorized for use in FDA Class III (or similar life-critical medical equipment) unless authorized officers of the parties have executed a special agreement specifically governing such use.

Only those TI components which TI has specifically designated as military grade or “enhanced plastic” are designed and intended for use in military/aerospace applications or environments. Buyer acknowledges and agrees that any military or aerospace use of TI components which have ***not*** been so designated is solely at the Buyer's risk, and that Buyer is solely responsible for compliance with all legal and regulatory requirements in connection with such use.

TI has specifically designated certain components as meeting ISO/TS16949 requirements, mainly for automotive use. In any case of use of non-designated products, TI will not be responsible for any failure to meet ISO/TS16949.

Products		**Applications**	
Audio	www.ti.com/audio	Automotive and Transportation	www.ti.com/automotive
Amplifiers	amplifier.ti.com	Communications and Telecom	www.ti.com/communications
Data Converters	dataconverter.ti.com	Computers and Peripherals	www.ti.com/computers
DLP® Products	www.dlp.com	Consumer Electronics	www.ti.com/consumer-apps
DSP	dsp.ti.com	Energy and Lighting	www.ti.com/energy
Clocks and Timers	www.ti.com/clocks	Industrial	www.ti.com/industrial
Interface	interface.ti.com	Medical	www.ti.com/medical
Logic	logic.ti.com	Security	www.ti.com/security
Power Mgmt	power.ti.com	Space, Avionics and Defense	www.ti.com/space-avionics-defense
Microcontrollers	microcontroller.ti.com	Video and Imaging	www.ti.com/video
RFID	www.ti-rfid.com		
OMAP Applications Processors	www.ti.com/omap	**TI E2E Community**	e2e.ti.com
Wireless Connectivity	www.ti.com/wirelessconnectivity		

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265