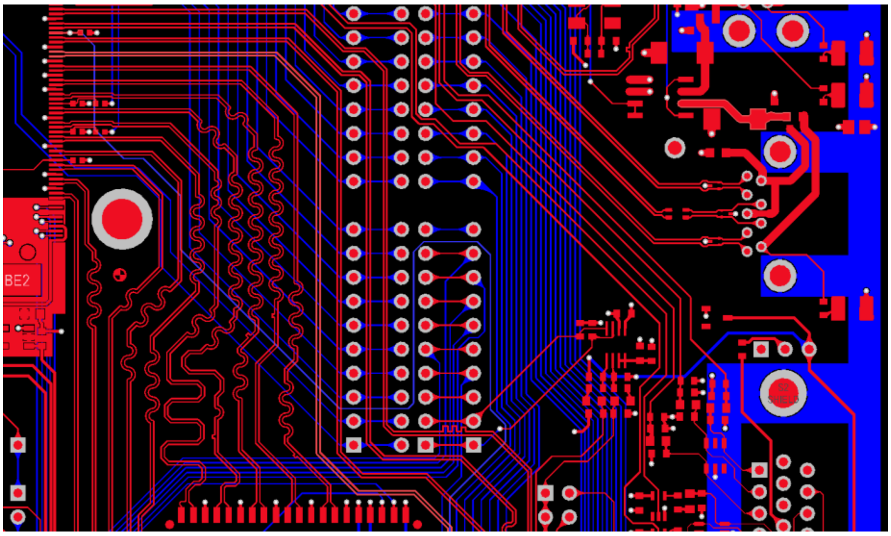
Swiss. Embedded. Computing.

Layout Design Guide



Issued by:	Toradex	**Document Type:**	Design Guide

Purpose:	This document is a guideline for designing a carrier board with high speed signals that is used with Toradex Computer Modules.

Document Version:	1.0

Revision History Date	Version	Remarks
14 April 2015	V1.0	Initial Release, based on section 2 of the Apalis Carrier Board Design Guide V1.0

1 Introduction

1.1 Overview

The latest Toradex Computer modules features new high speed interfaces such as PCI Express, SATA, HDMI, USB 3.0, Ethernet, and LVDS which require special layout considerations regarding trace impedance and length matching. Improper routing of such signals is a common pitfall in the design of an Apalis or Colibri carrier board. This document helps avoiding layout problems that can cause signal quality or EMC problems. Please read this document very carefully before you start designing a carrier board.

Please use this document together with the design guide of the appropriate Toradex computer module family and the datasheet of the module.

1.2 Additional Documents

1.2.1 Apalis Carrier Board Design Guide

This document provides additional information to the schematic design of a carrier board for the Apalis modules. It contains reference schematics, descriptions of the power architecture and information pertaining to the mechanical requirements of the module.

http://developer.toradex.com/hardware-resources/arm-family/carrier-board-design

1.2.2 Apalis Module Datasheets

There is a datasheet available for every Apalis Module. Amongst other things, this document describes the type-specific interfaces and the secondary function of the pins. Before starting the development of a customized carrier board, please check in this document whether the required interfaces are really available on the selected modules.

https://www.toradex.com/products/apalis-arm-computer-modules

1.2.3 Apalis Module Definition

This document describes the Apalis Module standard. It provides additional information about the interfaces.

http://docs.toradex.com/100240-apalis-module-specification.pdf

1.2.4 Colibri Carrier Board Design Guide

This document provides additional information about the schematic designs of carrier boards for the Colibri modules. It contains reference schematics, description about the power architecture and information related to the mechanical requirements of the module.

http://developer.toradex.com/hardware-resources/arm-family/carrier-board-design

1.2.5 Colibri Module Datasheets

There is a datasheet available for every Colibri Module. Amongst other things, this document describes the additional interfaces and the secondary function of the pins. Before starting the development of a customized carrier board, please refer this document to check if the required interfaces are really available on the selected modules.

https://www.toradex.com/products/colibri-arm-computer-modules

1.2.6 Toradex Developer Centre

You can find a lot of additional information at the Toradex Developer Centre, which is updated with the latest product support information on a regular basis.

Please note that the Developer Centre is common for all Toradex products. You should always check to ensure if the information is valid or relevant for your specific module.

http://www.developer.toradex.com

1.2.7 Carrier Board Design information

We provide the complete schematics and the Altium project file for Apalis and Colibri Evaluation Boards for free. This is a great help when designing your own Carrier Board.

http://developer.toradex.com/hardware-resources/arm-family/carrier-board-design

1.3 Abbreviations

Abbreviation	Explanation
ADC	Analogue to Digital Converter
AGND	Analogue Ground - separate ground for analogue signals
Auto-MDIX	Automatically Medium Dependent Interface Crossing - a PHY with Auto-MDIX is able to detect whether RX and TX need to be crossed (MDI or MDIX)
CAD	Computer-Aided Design, in this document is referred to PCB Layout tools
CAN	Controller Area Network - a bus that is manly used in automotive and industrial environment
CDMA	Code Division Multiplex Access - an abbreviation often used for a mobile phone standard for data communication
CEC	Consumer Electronic Control - a HDMI feature that allows to control CEC compatible devices
CPU	Central Processing Unit
CSI	Camera Serial Interface
DAC	Digital to Analogue Converter
DDC	Display Data Channel - an interface for reading out the capability of a monitor, in this document DDC2B (based on I^2C) is always meant
DRC	Design Rule Check - a tool for checking whether all design rules are satisfied in a CAD tool
DSI	Display Serial Interface
DVI	Digital Visual Interface. Digital signals are electrical compatible with HDMI
DVI-A	Digital Visual Interface Analogue only. Signals are compatible with VGA
DVI-D	Digital Visual Interface Digital only. Signals are electrical compatible with HDMI
DVI-I	Digital Visual Interface Integrated. Combines digital and analogue video signals in one connector
EDA	Electronic Design Automation - software for schematic capture and PCB layout (CAD or ECAD)
EDID	Extended Display Identification Data - timing setting information provided by the display in a PROM
EMC	Electromagnetic Compatibility -theory of unintentional generation, propagation, and reception of electromagnetic energy
EMI	Electromagnetic Interference - high frequency disturbances
eMMC	Embedded Multi Media Card - flash memory combined with MMC interface controller in a BGA package, used as internal flash memory

Abbreviation	Explanation
ESD	Electrostatic Discharge - high voltage spike or spark that can damage electrostatic-sensitive devices
FPD-Link	Flat Panel Display Link - high-speed serial interface for liquid crystal displays. In this document, also called LVDS interface.
GBE	Gigabit Ethernet - Ethernet interface with a maximum data rate of 1000Mbit/s
GND	Ground
GPIO	General Purpose Input/Output pin that can be configured to be either an input or output
GSM	Global System for Mobile Communications
HDA	High Definition Audio (HD Audio) - digital audio interface between CPU and audio codec
HDCP	High-Bandwidth Digital Content Protection - a copy protection system that is used by HDMI besides others
HDMI	High-Definition Multimedia Interface - it combines audio and video signal for connecting monitors, TV sets or Projectors, electrical compatible with DVI-D
I^2C	Inter-Integrated Circuit- a two wire interface for connecting low speed peripherals
I^2S	Integrated Interchip Sound- a serial bus for connecting PCM audio data between two devices
IrDA	Infrared Data Association - an infrared interface for connecting peripherals
JTAG	Joint Test Action Group - widely used debug interface
LCD	Liquid Crystal Display
LSB	Least Significant Bit
LVDS	Low-Voltage Differential Signaling, electrical interface standard that can transport very high speed signals over twisted-pair cables. Many interfaces like PCIe or SATA use this interface. Since the first successful application was the Flat Panel Display Link, LVDS became a synonymous for this interface. In this document, the term LVDS is used for the FPD-Link interface.
MIPI	Mobile Industry Processor Interface Alliance
MDI	Medium Dependent Interface, physical interface between Ethernet PHY and cable connector
MDIX	Medium Dependent Interface Crossed, an MDI interface with crossed RX and TX interfaces
mini PCIe	PCI Express Mini Card, card form factor for internal peripherals. The interface features PCIe and USB 2.0 connectivity
MMC	MultiMediaCard, flash memory card
MSB	Most Significant Bit
mSATA	Mini-SATA - a standardized form factor for small solid state drive, similar dimensions as mini PCIe
MXM3	Mobile PCI Express Module (second generation) - graphic card standard for mobile device. The Apalis form factor uses the physical connector but not the pin-out and the PCB dimensions of the MXM3 standard.
N/A	Not Available
N/C	Not Connected
OD	Open Drain
OTG	USB On-The-Go - a USB host interface that can also act as USB client when connected to another host interface
OWR	One Wire (1-Wire) - low speed interface which needs just one data wire plus ground
PCB	Printed Circuit Board
PCI	Peripheral Component Interconnect - parallel computer expansion bus for connecting peripherals
PCIe	PCI Express - high-speed serial computer expansion bus that replaces the PCI bus
PCM	Pulse-Code Modulation - digitally representation of analogue signals. Standard interface for digital audio
PD	Pull Down Resistor
PHY	Physical Layer of the OSI model
PMIC	Power Management IC, integrated circuit that manages amongst others the power sequence of a system
PU	Pull-Up Resistor
PWM	Pulse-Width Modulation
RGB	Red Green Blue - color channels in common display interfaces
RJ45	Registered Jack - a common name for the 8P8C modular connector that is used for Ethernet wiring
RS232	Single ended serial port interface
RS422	Differential signaling serial port interface, full duplex
RS485	Differential signaling serial port interface, half duplex, multi drop configuration possible
R-UIM	Removable User Identity Module - identifications card for CDMA phones and networks, an extension of the GSM SIM card

Abbreviation	Explanation
S/PDIF	Sony/Philips Digital Interconnect Format - optical or coaxial interface for audio signals
SATA	Serial ATA - high speed differential signaling interface for hard drives and SSD
SD	Secure Digital - flash memory card
SDIO	Secure Digital Input Output - an external bus for peripherals that uses the SD interface
SIM	Subscriber Identification Module - identification card for GSM phones
SMBus	System Management Bus (SMB) -, two wire bus based on the I^2C specifications, used specially in x86 design for system management.
SoC	System on a Chip - IC which integrates the main component of a computer on a single chip
SPI	Serial Peripheral Interface Bus - synchronous four wire full duplex bus for peripherals
TIM	Thermal Interface Material - thermal conductive material between CPU and heat spreader or heat sink
TMDS	Transition-Minimized Differential Signaling - serial high speed transmitting technology that is used by DVI and HDMI
TVS Diode	Transient-Voltage-Suppression Diode - diode that is used to protect interfaces against voltage spikes
UART	Universal Asynchronous Receiver/Transmitter - serial interface, in combination with a transceiver a RS232, RS422, RS485, IrDA or similar interface can be achieved
USB	Universal Serial Bus - serial interface for internal and external peripherals
VCC	Positive supply voltage
VGA	Video Graphics Array - analogue video interface for monitors

Table 1: Abbreviations

2 General Considerations

The Apalis and Colibri modules feature a range of high speed interfaces which need special treatment with regards to its PCB layout. This section describes a collection of basic rules to follow. It should be noted however that it is not often possible to follow all the rules. It is the job of the design engineer, with the aid of this design guide, to decide which rules can be violated, in what area, for which signals, and when it is necessary to do so.

The interfaces have an 'importance priority' over one another when it comes to ensuring optimal routing of designs. The below-mentioned list describes the importance priority of the signals. PCIe is the first one on the list and has the highest priority, and should be routed with special care. Signals continue to be ordered with descending priority and as such become less problematic with respect to layout and routing. Often, a good approach to take is to layout and route interfaces in order of their importance priority, from high to low.

1. PCI Express
2. USB 3.0 (Super Speed signals)
3. SATA
4. Ethernet
5. HDMI
6. LVDS Display
7. USB 2.0
8. SD/MMC/SDIO
9. Parallel RGB LCD Interface
10. Parallel Camera Input
11. HD Audio
12. Analogue VGA
13. Analogue Audio, ADC Inputs, Touch Panel
14. Low Speed Interfaces (I2C, UART, SPI, CAN, PWM, OWR, S/PDIF, Keypad, GPIO)

3 PCB Stack-Up

In order to reduce reflections at high speed signals, it is necessary to match the impedance between source, sink, and transmission line. The impedance of a signal trace depends on its geometry and its position with respect to any reference planes. The trace width and spacing between differential pairs for a specific impedance requirement is dependent on the chosen PCB stack-up. As there are limitations in the minimum trace width and spacing which depends on the type of PCB technology and cost requirements, a PCB stack-up needs to be chosen which allows all the required impedances to be realized. The presented stack-ups in the following subsections are intended as examples which can be used as a starting point for helping in stackup evaluation and selection. If a different stack-up is required other than those shown in the examples, please recalculate the dimensions of the traces. Work closely with your PCB manufacturer when selecting suitable stack-up solution.

3.1 Four Layer Stack–Up

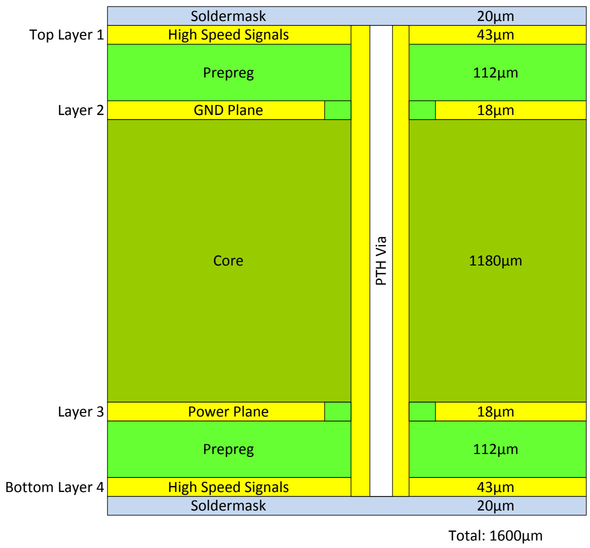


Figure 1: Four Layer PCB Stack-Up Example

The high speed signals on the top layer are referenced to the ground plane on layer 2. Since the references for the high-speed signals on the bottom layer are the power planes on Layer 3, it is necessary to place stitching capacitors between the aforementioned power planes and ground. More information about stitching capacitors can be found in section 6.8. In this stack-up, it is preferential to route high speed signals on the top layer as opposed to the bottom layer so that the signals have a direct reference to the ground layer. For some designs it may be desirable to have the bottom layer as primary high speed routing layer. In this case, the power and ground usage on Layer 2 and 3 could be swapped.

3.2 Six Layer Stack–Up

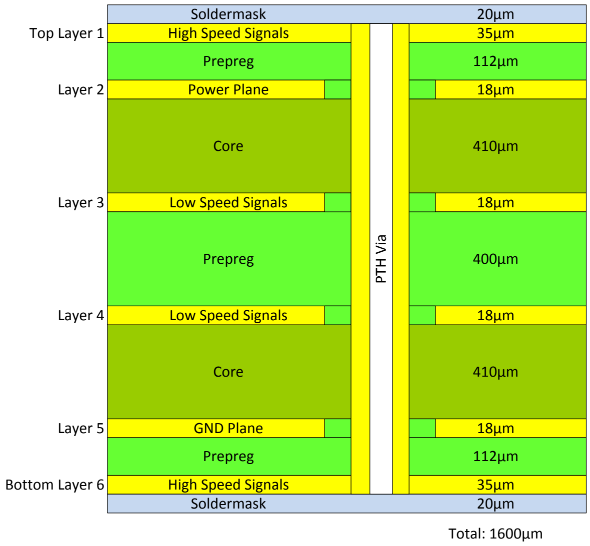


Figure 2: Six Layer PCB Stack-Up Example

In this example, the reference planes for the high-speed signals on the top layer are the power planes on layer 2. Stitching capacitors from their associated reference power planes to the ground is therefore required. More information about stitching capacitors can be found in section 6.8. The signal reference for the bottom layer is the ground plane on layer 5. In this stack-up, it is preferable to route high-speed signals on the bottom layer. As in the previous example, power and ground layers could be swapped if it is desirable to have the primary high-speed routing layer on the top layer.

The reference planes for signals on Layer 3 are located on Layer 2 and 5. The same reference planes are used by signals routed on Layer 4. As the reference planes are on layers which have a relatively large distance from Signal Layers 3 and 4, the traces would need to be very wide in order to achieve a common impedance of 50Ω. Therefore, these layers are not suitable for routing high-speed signals. In this stack-up approach, Layers 3 and 4 can only be used for routing low-speed signals where impedance matching is not required.

3.3 Eight Layer Stack–Up

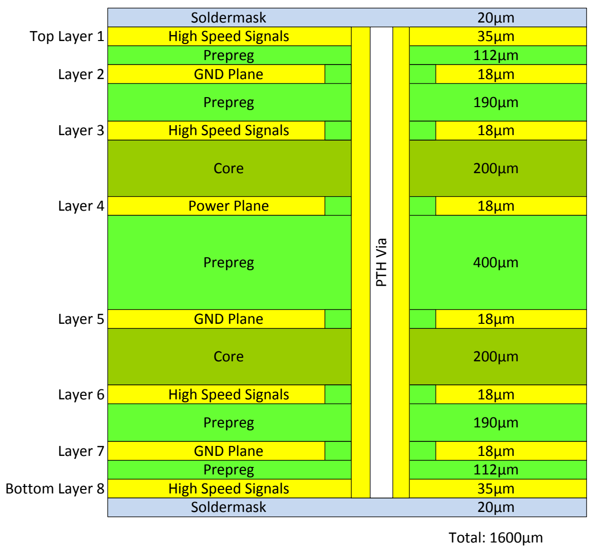


Figure 3: Eight Layer PCB Stack-Up Example

The signals on the top layer are referenced to the plane in Layer 2, while the signals on the bottom layer are referenced to layer 7. The reference planes for Signal Layer 3 are the ground plane on Layer 2 and the power planes on Layer 4. When routing high-speed signals on Layer 3, stitching capacitors need to be placed between the power and the ground planes. The power planes on Layer 5 and 7 are used as references for the high-speed signals routed on Layer 6.

The inner layer 6 with the two adjacent ground planes is the best choice for routing high-speed signals which have the most critical impedance control requirements. The inner layers cause less EMC problems as they are capsulated by the adjacent ground planes. As Layer 3 is referenced to a power plane, outer layer 1 and 8 are preferable for high-speed routing if Layer 6 is already occupied.

4 Trace Impedance

Care should be taken to distinguish between single-ended and differential trace impedance. High-speed single ended signals such as the parallel RGB LCD or camera interface need to be routed with the specified single-ended impedance. This is the impedance between the trace and the reference ground.

High-speed differential pair signals such as PCIe, SATA, USB, HDMI etc. need to be routed with differential impedance. This is the impedance between the two signal traces of a pair. As the signals are also referenced to ground, each differential pair signal also has single-ended impedance. When selecting trace geometry, priority should be given to matching the differential impedance over the single-ended impedance. The differential impedance is always smaller than twice the single ended impedance

$$Z_{Differntial} < 2 \cdot Z_{Single\ Ended}$$

The signals allow a certain impedance tolerance (e.g. 50Ω ±15%). When defining trace geometry, try to keep the calculated impedance value as close as possible to the exact impedance value. This allows greater flexibility during PCB manufacture. Variation in impedances will occur between different production lots. If the calculated impedance is in the middle of the tolerance band, it will help ensure the maximum production yield.

Different tools can be used for calculating the trace impedance. Polar Instruments offers a widely used tool. Many PCB manufacturers use this tool. PCB manufacturers can often help customers with impedance calculations, and it is suggested that you work with your chosen PCB manufacturer during your design. Many PCB layout tools offer a very basic impedance calculator. Unfortunately, these calculators are not reliable for all situations.

Traces on the top or bottom layer have only one reference plane. These traces are called Microstrip. The following figure shows the geometry of such Microstrips. H1 is the distance from the trace to the according reference plane. Er1 is the relative permittivity of the isolation material. The traces have a trapezoid form due to the etching process. In the layout tool, the traces have to be designed with a width of W1. W2 depends on the trace height (T1) and the duration of the etching. Contact your PCB manufacturer in order to get the information about the resulting width W2. S1 is the spacing within a differential pair.

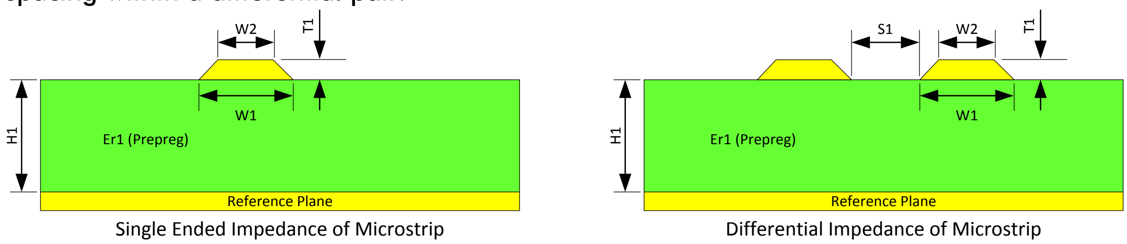
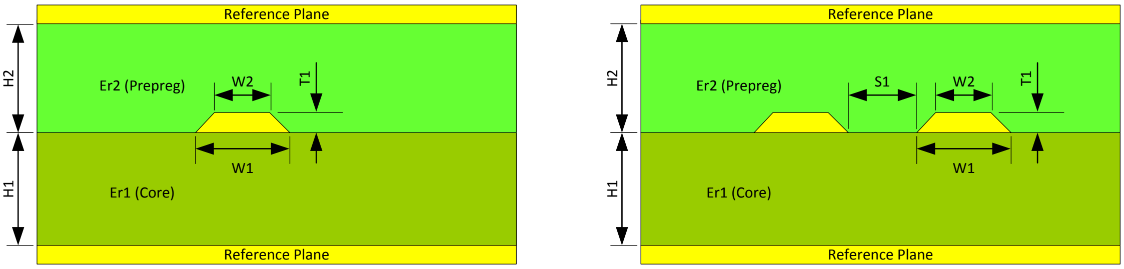


Figure 4: Trace Geometry of Microstrips

Traces in the inner layer of a PCB have two reference planes, reducing electromagnetic emissions and increasing immunity to external noise sources. These traces are called striplines. The following figure shows the geometry of such striplines. When making impedance calculation of striplines, special care needs to be taken when it comes to the isolation thickness H1 and H2. H1 is the thickness of the core material. The traces are embedded in the prepreg material. As the traces have a finite height, the prepreg height H2 depends on the copper density. The relative permittivity of the core and prepreg material can be slightly different. Many impedance calculation tools can take this in account.



Single Ended Impedance of Stripline   Differential Impedance of Stripline

Figure 5: Trace Geometry of Striplines

The following table shows typical trace geometries for traces using the four layer stack-up, presented in section 3.

Signal Type	Required $Z_{Single\ Ended}$	Required $Z_{Differential}$	Layer	Trace Type	W1	S1
Common Single Ended	50Ω	N/A	1, 4	Microstrip	180µm	N/A
PCIe, USB, HDMI,	50Ω	90Ω	1, 4	Microstrip	180µm	190µm
SATA	55Ω	90Ω	1, 4	Microstrip	150µm	150µm
Ethernet	55Ω	95Ω	1, 4	Microstrip	150µm	165µm
LVDS	55Ω	100Ω	1, 4	Microstrip	150µm	200µm

Table 2: Four Layer Stack-Up Example

The following table shows typical trace geometries for traces using the six layer stack-up, presented in section 3.

Signal Type	Required $Z_{Single\ Ended}$	Required $Z_{Differential}$	Layer	Trace Type	W1	S1
Common Single Ended	50Ω	N/A	1, 6	Microstrip	180µm	N/A
PCIe, USB, HDMI,	50Ω	90Ω	1, 6	Microstrip	180µm	200µm
SATA	55Ω	90Ω	1, 6	Microstrip	150µm	130µm
Ethernet	55Ω	95Ω	1, 6	Microstrip	150µm	165µm
LVDS	55Ω	100Ω	1, 6	Microstrip	150µm	200µm

Table 3: Six Layer Stack-Up Example

The following table shows typical trace geometries for traces used in the eight layer stack-up, presented in section 3.

Signal Type	Required $Z_{Single\ Ended}$	Required $Z_{Differential}$	Layer	Trace Type	W1	S1
Common Single Ended	50Ω	N/A	1, 8	Microstrip	180µm	N/A
			3, 6	Stripline	170µm	N/A
PCIe, USB, HDMI,	50Ω	90Ω	1, 8	Microstrip	180µm	200µm
			3, 6	Stripline	170µm	185µm
SATA	55Ω	90Ω	1, 8	Microstrip	150µm	130µm
			3, 6	Stripline	140µm	120µm
Ethernet	55Ω	95Ω	1, 8	Microstrip	150µm	165µm
			3, 6	Stripline	140µm	155µm
LVDS	55Ω	100Ω	1, 8	Microstrip	150µm	200µm
			3, 6	Stripline	140µm	200µm

Table 4: Eight Layer Stack-Up Example

5 Component Placement and Schematic Optimizations

The placement of the components is very often an underestimated subtask of the PCB design. Problems with signal return paths (see section 6.8) are very often related to suboptimal placement of the components. If a signal needs to cross a splitting of its reference plane, one should first check whether this splitting is really unavoidable. Quite often, the solution can be a better placement of the components.

The high-speed signal connection should play a major role in the decisions that are taken during the placement. Certain signals only allow a certain amount of vias on the board. This means the number of layer changes should be kept as low as possible. When placing the components, try to avoid the need of crossing high-speed signals. Maybe, it would be worthwhile to place an interface chip on the bottom layer. Some interfaces, such as the PCIe, support polarity reversal. This means you can swap the positive and negative signal pins in order to avoid the need of complicatedly crossing the two traces of a differential pair signal.

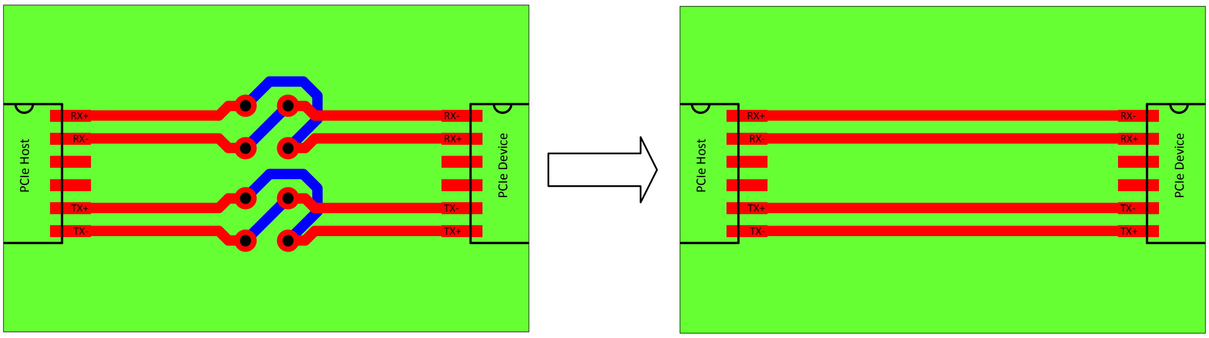


Figure 6: Make use of polarity reversal if supported by the interface

Some of multi-lane interfaces support the lane reversal. This allows avoiding the crossing of the lanes. Please note that while polarity reversal is a mandatory feature of the PCIe interface, the lane reversal depends on the peripheral devices and host controller of the computer module. Please read carefully the appropriate datasheets before reversing the lanes.

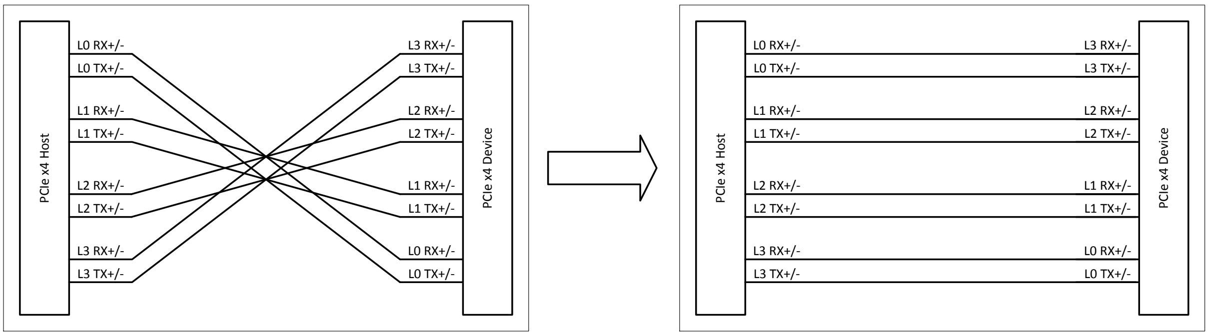


Figure 7: Make use of lane reversal if supported by the interface

When placing the components, take in account that high-speed signals normally need to have more spacing than other signals. If there are different type of signals involved, like analogue signals and high-speed digital signals, placement of the components needs to be done very carefully. The need of a complicated reference plane shape is often a good indication that the component placement is suboptimal. Do not be afraid of doing placement and routing iterations.

6 High-Speed Layout Considerations

6.1 Power Supply

Digital circuits often draw a non-continuous current from their supply power. Peak current consumption can be relatively large with high frequency components. If the supply traces are long, such current peaks can cause high frequency noise emission, which can be introduced into other signals. As traces have parasitic resistance and inductance, this high frequency noise can be coupled into supplies for other circuits (see left figure below). Another problem is that the parasitic inductance of the supply trace reduces the ability for the trace to carry the current peaks, which can cause voltage drops at the consuming circuit. It is therefore necessary to add bypass capacitors to the power input pins of digital circuits, which act to provide a reservoir of energy that can be drawn on to help supply the short term peak currents that may be required.

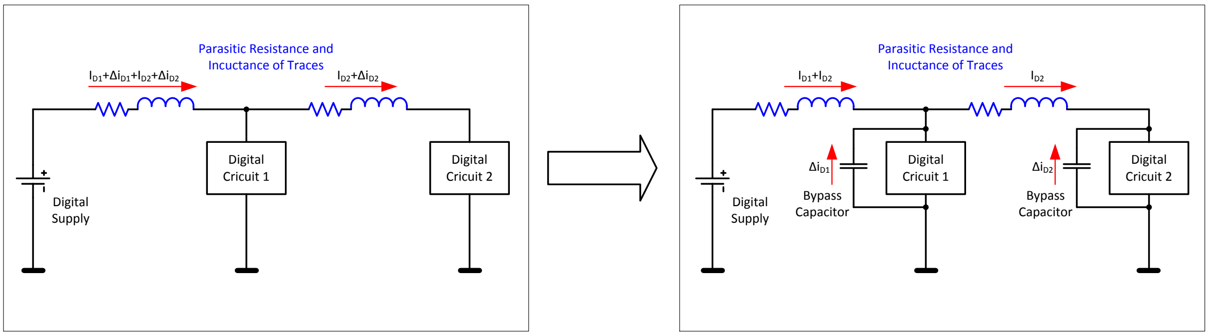


Figure 8: Add bypass capacitors

If possible, individual bypass capacitors should be placed on every power supply pin of an integrated circuit. If supply pins are close together, a bypass capacitor may be shared between both pins. Capacitors should be placed as close as possible to supply pins. Try to enlarge the supply trace widths. Try to keep the traces short. Current flow direction should also be considered. It is preferable that the current first passes the bypass capacitor and then enters the supply pin. Add an adequate amount of vias to the power supply traces. As a rule of thumb, place one via for one ampere of current consumption. If the decoupling capacitors are placed on the other side of the PCB and the current needs to go through vias, also consider the peak current. Also, do not forget about the ground return current. The ground should have at least the same amount of vias as the supply.

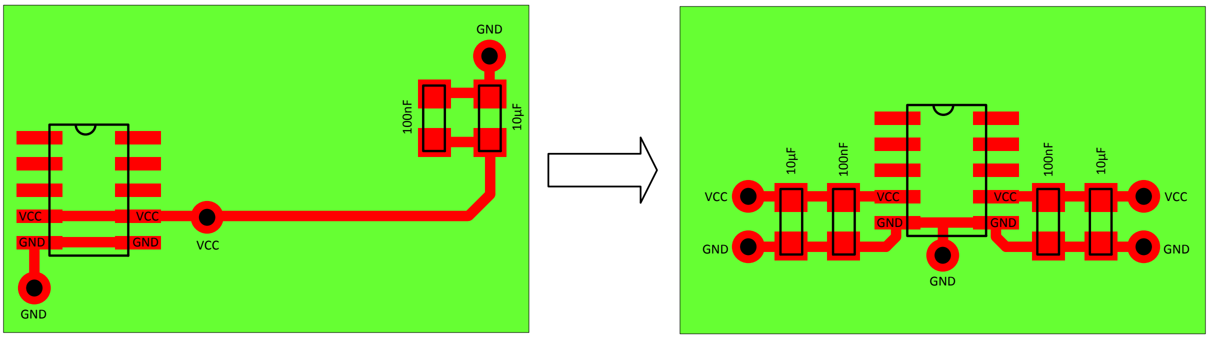


Figure 9: Place Bypass Capacitors close to IC pins

Large capacitors have a limitation in the speed they can provide energy to counter the current peaks, while small capacitors may not have enough capacity to satisfy the energy demand. Therefore, often a combination of small and big (e.g. 100nF + 10 µF) capacitors is a good choice for such supply circuits. Ensure that the smaller capacitor is placed closer to the supply pin than the bigger one.

If buck or boost converters are used on the baseboard, make sure that its layout follows the recommendations of the supplier.

Be aware of the total capacity on a voltage rail while switching the voltage. If the rails are switched on too fast, the current peaks for charging all the bypass capacitors can be very high. This can produce unacceptable disturbances (EMI) or can trigger an overcurrent in the protection circuit. Maybe the switching speed needs to be limited. The following figure shows a simple voltage rail switch circuit. C1 and R1 limit the switching speed. The values need to be optimized according to the requirements. It is recommended to place a bypass capacitor (C2) close to the switching transistor.

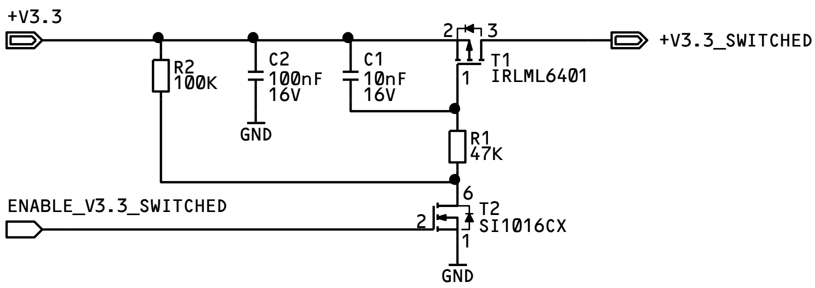


Figure 10: Simple Voltage Switch Circuit

When routing power traces, always be aware of the electrical resistance and inductivity. Try to make the traces as wide as possible by preferably using power planes instead of traces. Be aware of the copper thickness of the traces. A common specification for copper foils is half ounce of copper per square foot. This is equal to a thickness of 17μm. As a rule of thumb, the resistance of a square shaped trace has a resistance of 1mΩ. This means that a trace with width of 100μm has a resistance of 1mΩ per 100μm length. In other words, a trace with the width of 100μm and a length of 100mm has a resistance of 1Ω.

Copper foils on the outer layers of a PCB (top and bottom layer) often are thicker due to the via plating process. A common value is one ounce of copper per square foot. This equals to a thickness of 35μm. The traces on such layers have half electrical resistance which is 0.5mΩ for a square shaped trace. Also, consider the electrical resistance of vias. A rule of thumbs is to place at least one via for every ampere of current.

Signal vias create voids in the power and ground planes. Improper placement of vias can create plane areas in which the current density is increased. These areas are also called hot spots. It is important to avoid these hot spots. Often a good approach is to place the vias in a grid that leaves enough space between the vias for the power plane to pass.

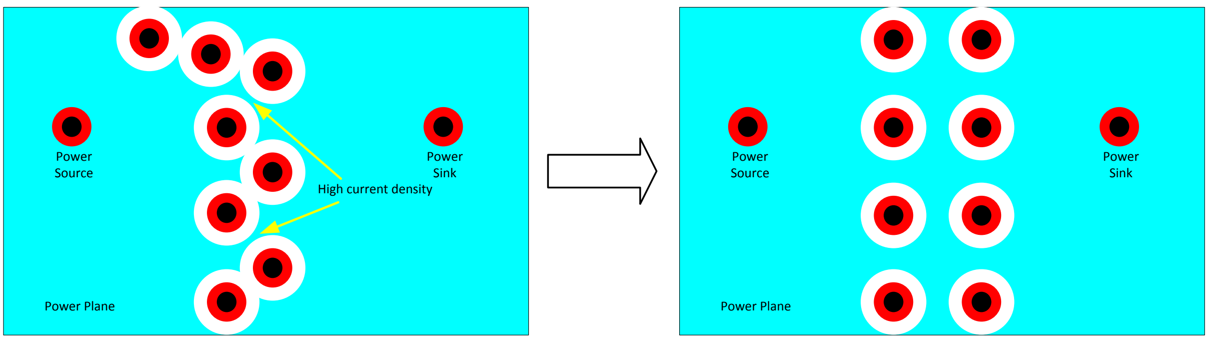


Figure 11: Avoid Copper Plane Hot Spots

6.2 Trace Bend Geometry

When routing high-speed signals, bends should be minimized. If bends are needed, use 135° bends instead of 90°.

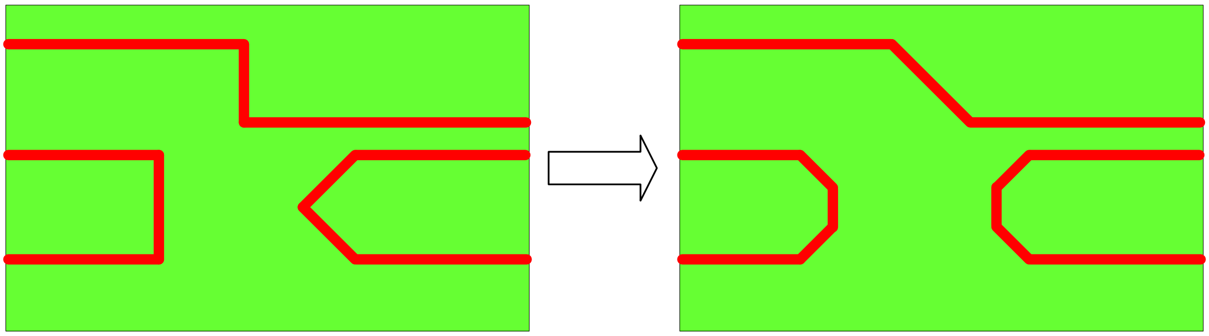


Figure 12: Use 135° Bends instead of 90°

Serpentine traces (also called meander) are often needed when a certain trace length needs to be achieved. Keep a minimum distance of four times the trace width between adjacent copper in a single trace. The individual segments of the bends should be at least 1.5 times the trace width. A lot of DRCs in CAD tools do not check these minimum distances as the traces are part of the same net.

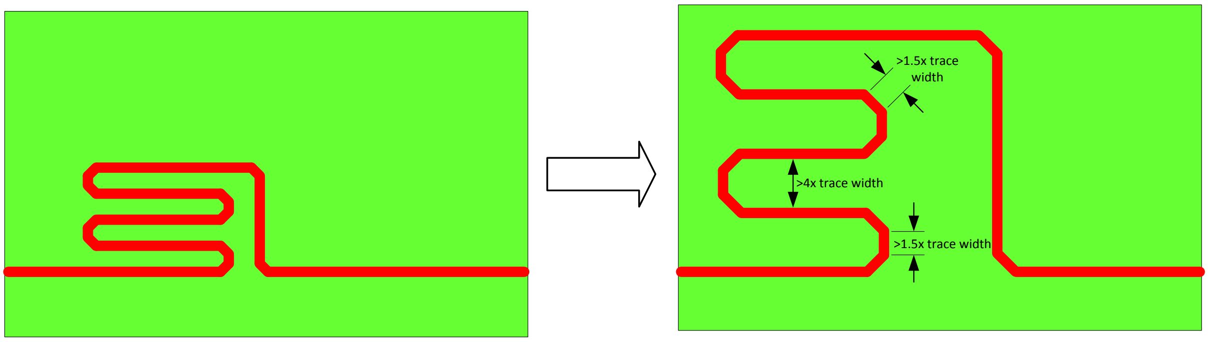


Figure 13: Keep minimum Distance and Segment Length at Bends

6.3 Signal Proximity

Information about the required minimum distance between high-speed signals can be found in section 0. A minimum distance is required in order to minimize crosstalk between traces. The level of crosstalk depends on the distance between two traces and the length in which they are closely routed. Sometimes, bottlenecks can force the routing of traces closer than to what is normally permitted. Try to minimize such areas and enlarge the distance between the signals outside the bottleneck. If there is space available, try to enlarge the distance between the high speed-signals (and between high-speed and low-speed signals) even if the minimum trace separation requirement has been met.

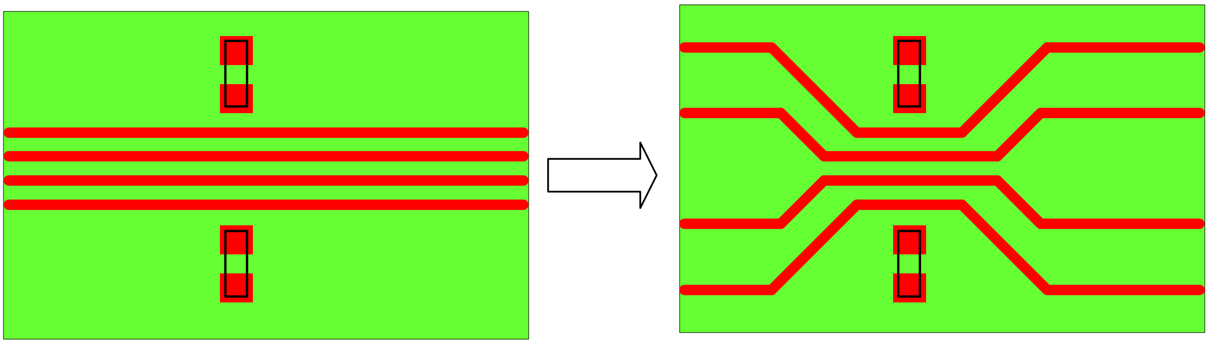


Figure 14: Try to increase Spacing between Traces whenever it is possible

6.4 Trace Stubs

Long stub traces can act as antennas and therefore increase problems complying with EMC standards. Stub traces can also produce reflections which negatively impacts signal integrity. Common sources for stubs are pull-up or pull-down resistors on high speed signals. If such resistors are required, route the signals as a daisy chain.

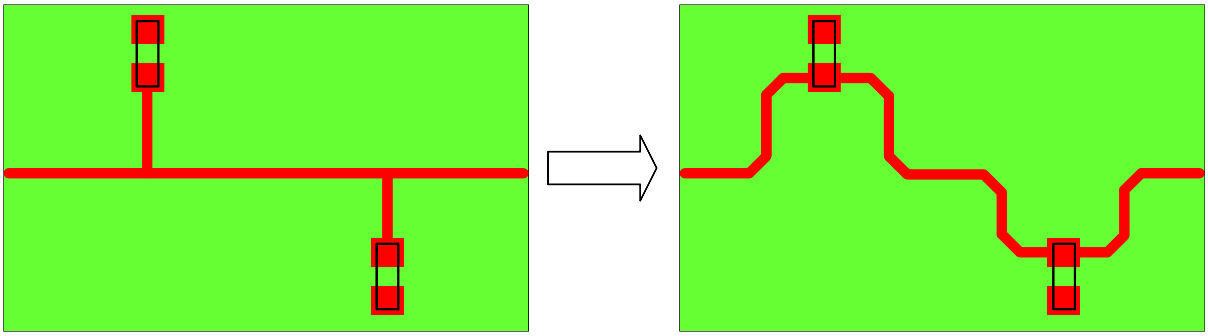


Figure 15: Avoid Stub Traces by Daisy Chain Routing

As a rule of thumb, stubs longer than a tenth of the wavelength should be considered as problematic. The following example shows the calculations on a Gen3 PCIe signal:

$$l_{MAX} \ll \frac{1}{10}\lambda_{MIN} = \frac{v}{10 \cdot f_{MAX}} = \frac{\frac{c}{\sqrt{\varepsilon_r}}}{10 \cdot f_{MAX}} = \frac{\frac{300 \cdot 10^6 \, m/_s}{\sqrt{4.5}}}{10 \cdot 4GHz} = 3.5mm$$

Vias can also act as stubs. For example, in a six layer board, when a signal changes from layer 1 to 3 by using a via, the via creates a stub which reaches layer 6. Back-drilling the vias in order to avoid such stubs is a quite expensive technology and one which is not supported by most PCB manufacturers. The only practical solution is to reduce the number of vias in high-speed traces.

6.5 Ground Planes under Pads

The impedance of a trace depends on its width and the distance between trace and reference plane. A wide trace has lower impedance than a thin one with the same distance. The same effect also exists for connector and component pads. A large pad has significantly lower impedance than the trace which is connected to the pad. This impedance discontinuity can cause reflections reduces signal integrity. Therefore, under large connector and component pads, a plane obstruct should be placed. In this case, an active reference plane should be placed on another layer. This reference plane needs to be stitched with vias to the normal reference plane.

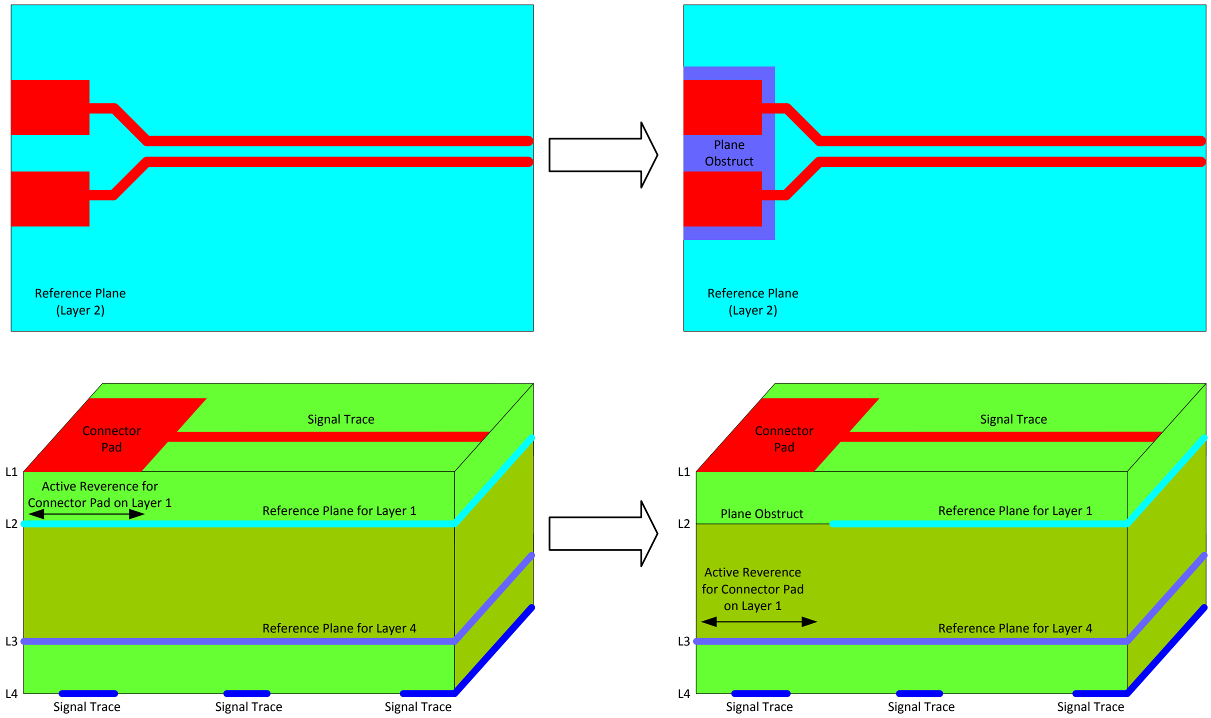


Figure 16: Remove Ground Plane under large Pads

Vias are another source of impedance discontinuity. In order to minimize the effect, the unused pads of vias in inner layers should be removed. This can be done at design time in the CAD tool or by the PCB manufacturer.

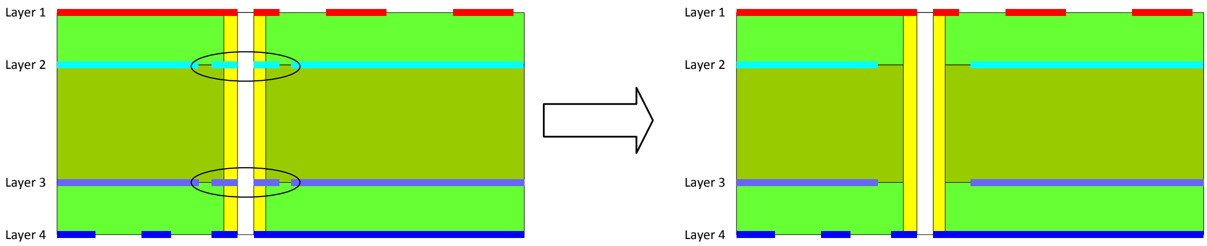


Figure 17: Remove unused Via Pads

6.6 Differential Pair Signals

High-speed differential pair signals need to be routed in parallel with a specific, constant distance between the two traces. This distance is required in order to obtain the specified differential impedance (see section 0). Differential pair signals need to be routed symmetrically. Try to minimize the area in which the specified spacing is enlarged due to pad entries.

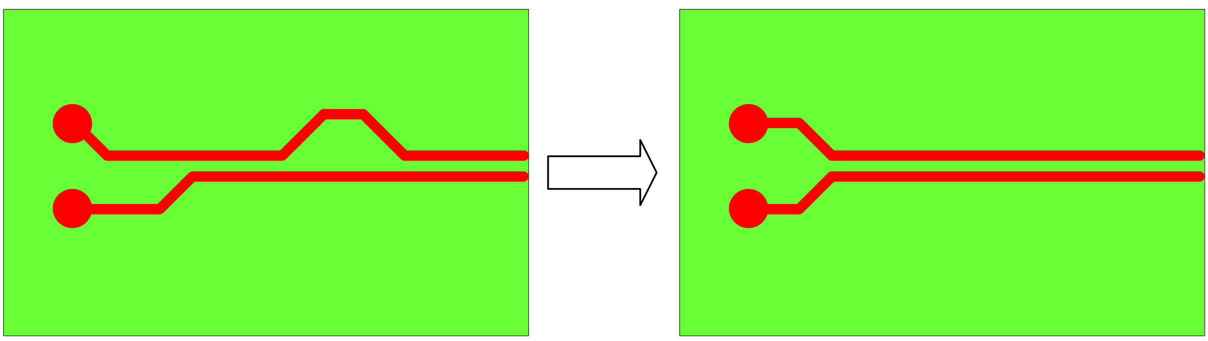


Figure 18: Route Differential Pairs symmetrically and keep Signals always parallel

It is not permitted to place any components or vias between the differential pairs, even if the signals are routed symmetrically. Components and vias between the pairs could lead to EMC compliance problems and create an impedance discontinuity.

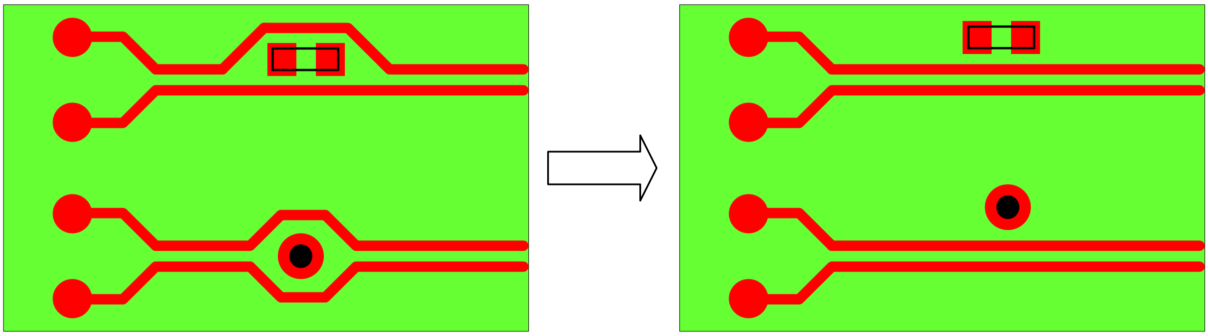


Figure 19: Do not put any Components or Vias between Differential Pairs

Some differential pair high-speed signals require serial coupling capacitors. Place such capacitors symmetrically. The capacitors and the pads create impedance discontinuities. 0402 sized capacitors are preferable, 0603 are acceptable. Do not place larger packages such as 0805 or C-packs.

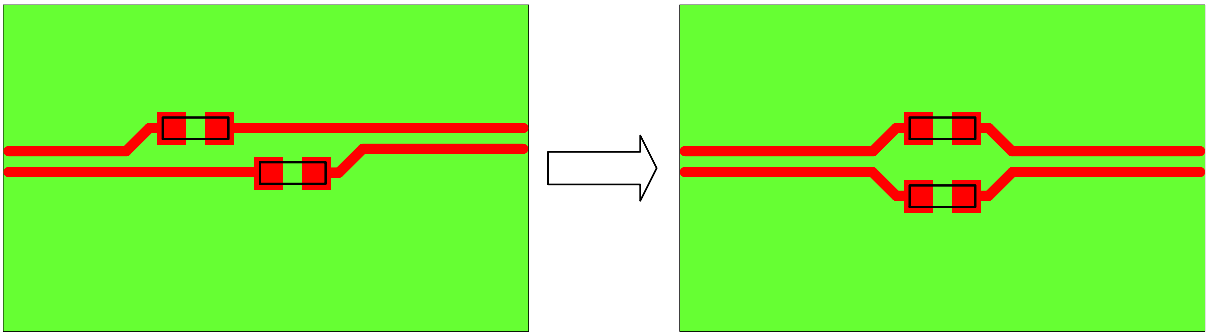


Figure 20: Place Coupling Capacitors symmetrical

Vias introduce a huge discontinuity in impedance. Try to reduce the amount of placed vias to a minimum and place the vias symmetrically.

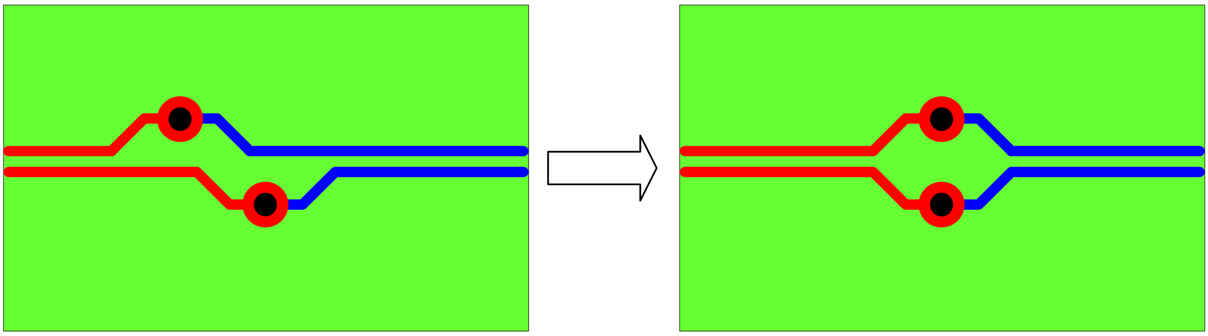


Figure 21: Place Vias symmetrical

In order to meet the impedance requirements of a differential pair, both signal traces need to be routed on the same layer. Add the same amount of vias to the traces.

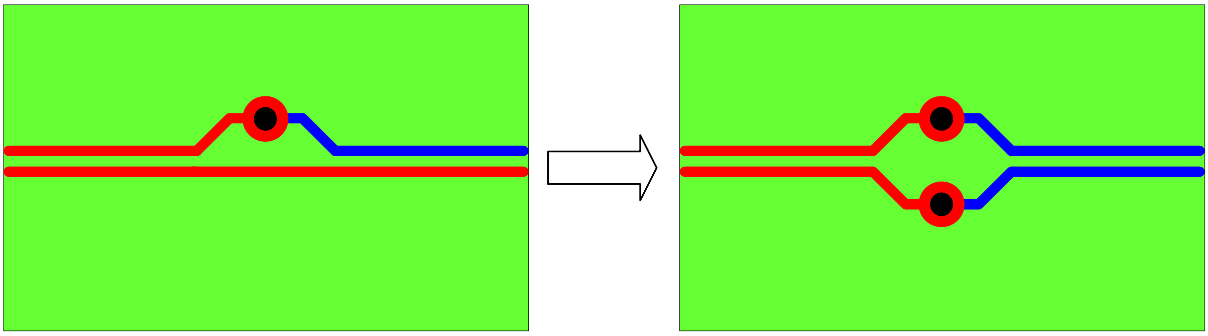


Figure 22: Route Pairs on the same Layer, place same Amount of Vias

6.7 Length Matching

High-speed interfaces have additional requirements regarding the time of arrival skew between different traces and pairs of signals. For example, in a high speed parallel bus, all data signals need to arrive within a time period in order to meet the setup and hold time requirements of the receiver. The carrier board designer needs to make sure that such permitted skew is not exceeded. In order to meet this requirement, length matching is required. In section 0, more information about the length matching requirements for each interface type can be found. Often, the requirements are given as a maximum time skew. In order to calculate the maximum trace length difference, the propagation speed on the PCB needs to be estimated. The following formula can be used for calculating the speed:

$$v = \frac{c}{\sqrt{\varepsilon_r}}$$

The symbol c stands for the speed of light while ε_r is the relative permittivity of material between the trace and the reference plane. The relative permittivity of an FR-4 PCB is around 4.5 while air has 1. A few of the coupling between Microstrip signals on the outer layer of a PCB and the reference plane is over the air and the solder mask. Since the relative permittivity of both materials is lower than the one of the FR-4, the signals are propagated slightly faster than in the inner layers (Striplines). As a rule of thumb, the signals on a PCB are propagated at half the speed of light. This equals to a speed of around 150 μm/ps.

$$v \approx 150 \frac{\mu m}{ps}$$

Differential pair signals often require a very tight delay skew between the positive and negative signal traces. Therefore, length differences need to be compensated for using serpentines (also called meanders). The geometry of serpentine traces need to be carefully chosen in order to reduce impedance discontinuity. The following figure shows the requirements for ideal serpentine traces:

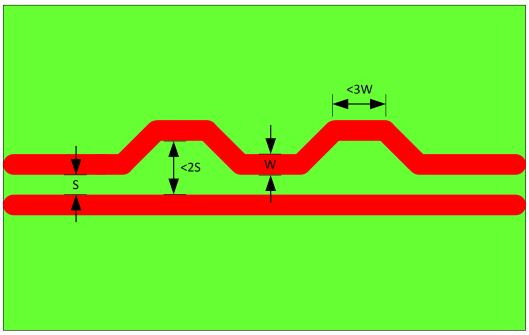


Figure 23: Preferred Serpentine Trace Geometry

The serpentine traces should be placed at the origin of the length mismatching. This ensures that the positive and negative signal components are propagated synchronously over the major part of the connection.

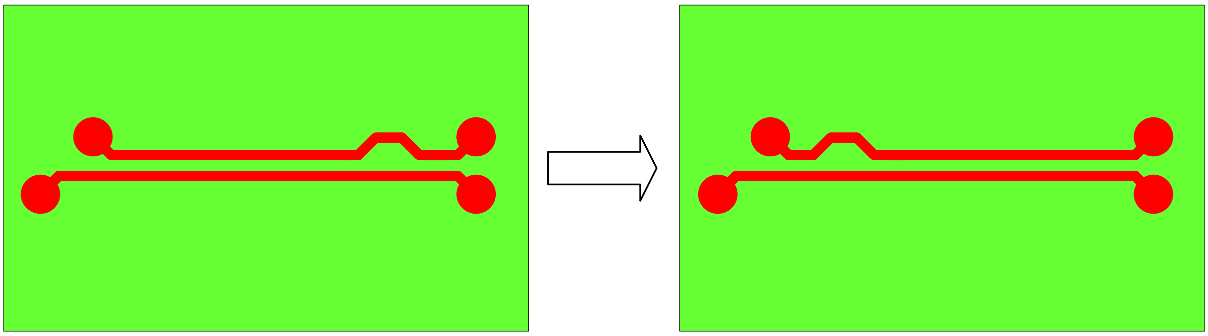


Figure 24: Add Length Correction to the Mismatching Point

Bends are a common source of length mismatching. The compensation should be placed close to the bend with a maximum distance of 15mm.

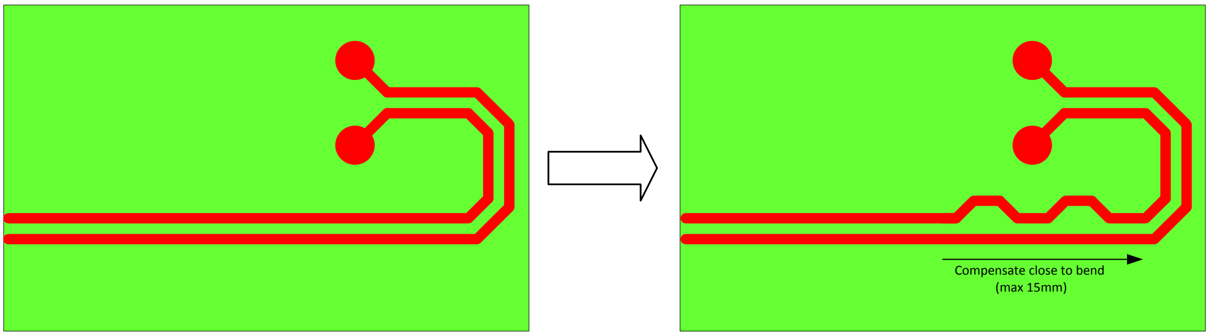


Figure 25: Place Length Compensation close to Bend

Often two bends compensate each other. If the bends are closer than 15mm, no additional compensation with serpentines is necessary. The signals should not propagate asynchronously over a distance greater than 15mm.

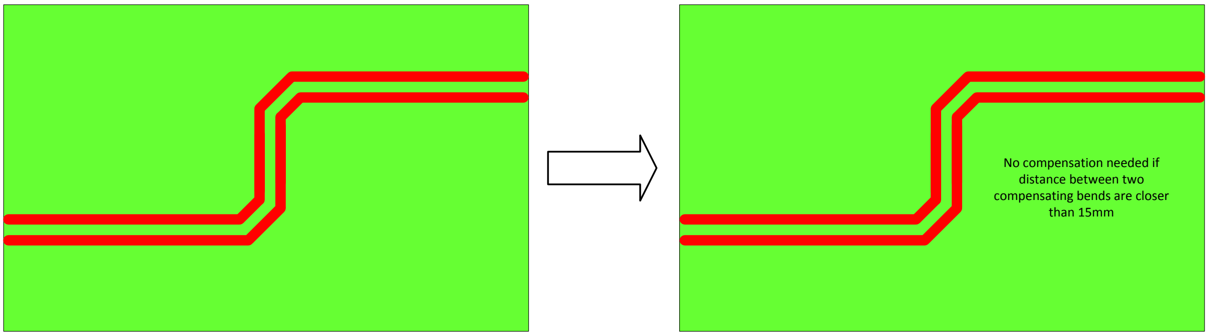


Figure 26: Bends can compensate each other

Each segment of a differential pair connection needs to be matched individually. A connection can be segmented by a connector, serial coupling capacitors or vias. The two bends in the following figure would compensate each other. Since the vias divide the differential pair into two segments, the bends need to be compensated individually. This makes sure that the positive and negative signals are propagated synchronously through the vias. The violation of this rule might need to be checked manually as the DRC may only check the length difference over the whole connection.

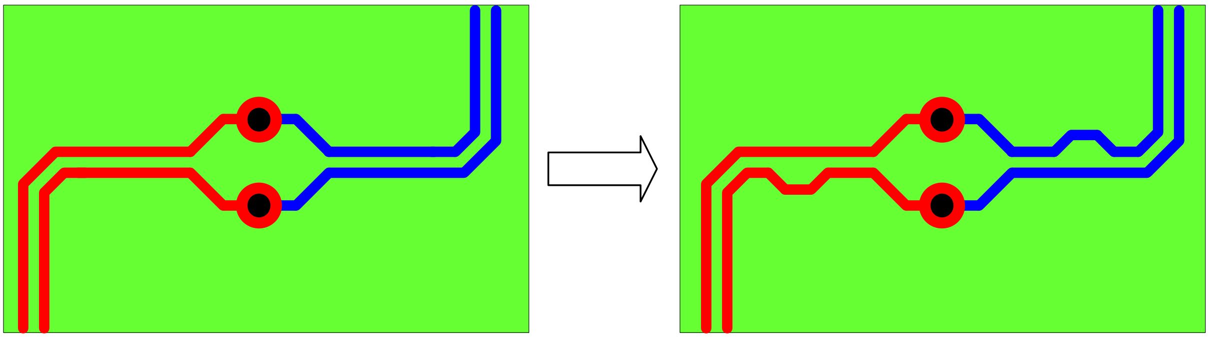


Figure 27: Length Differences need to be compensated in each Segment

The signal speed is not equal for different layers. Since the difference is hard to estimate, it is preferable to route signals on the same layer if they need to be matched. For example, the LVDS display interface requires tight matching between the signal pairs and the clock pair. It is preferable to route all data and clock signals of an LVDS channel on the same layer.

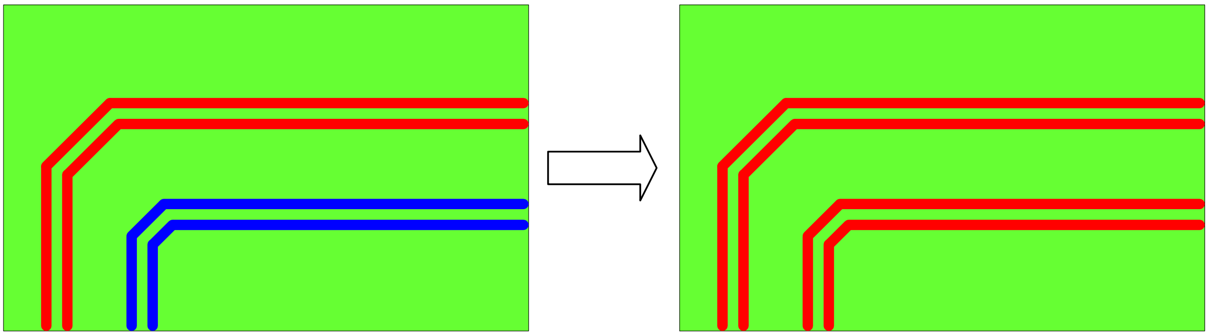


Figure 28: Pairs within same Interface should be routed preferable on same Layer

Please be aware that some CAD tools also accounts the trace length inside a pad to its total length. The following figure shows two layouts which are similar from an electrical point of view. On the left side, the traces inside the capacitor pads do not have equal length. Even though the signals are not using the internal traces, some CAD tools use this as part of the length calculation and show a length difference between the positive and negative signal. In order to minimize the impact of incorrect trace length calculations, ensure that the pad entry is equal for both signals. Similarly, some CAD tools do not take in account the length of vias when calculating the total length. As differential pairs should have the same amount of vias in both traces, the error does not affect the length matching. It can, however, affect calculations for matching two differential pairs or the matching of parallel buses.

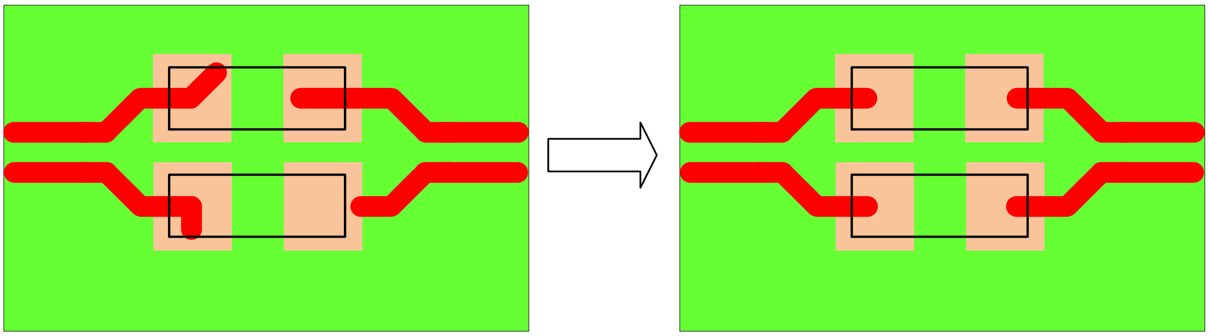


Figure 29: Length Calculation Problem of some CAD Tools

Whenever possible, a symmetric breakout of differential pair signal is preferred in order to avoid the need of serpentine traces.

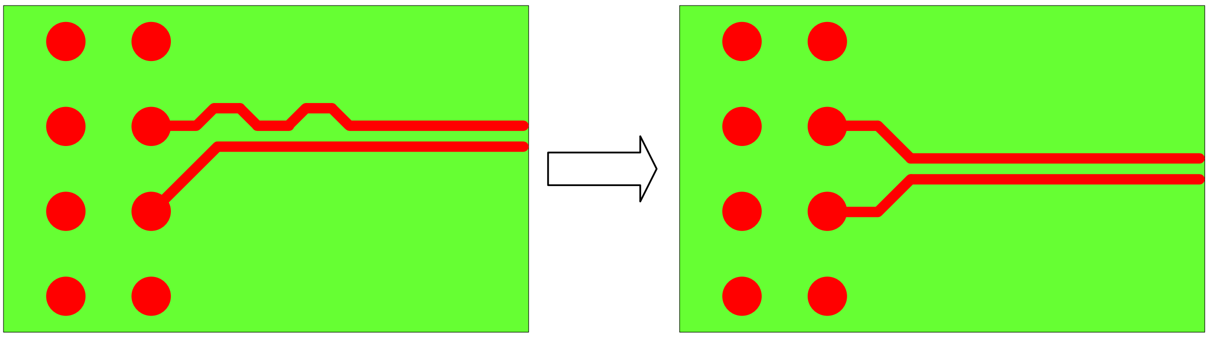


Figure 30: Preferred symmetrical Breakout

If the space between the pads permits, try to add a small loop to the shorter trace. This is the preferred solution for matching the length difference as opposed to creating a serpentine trace.

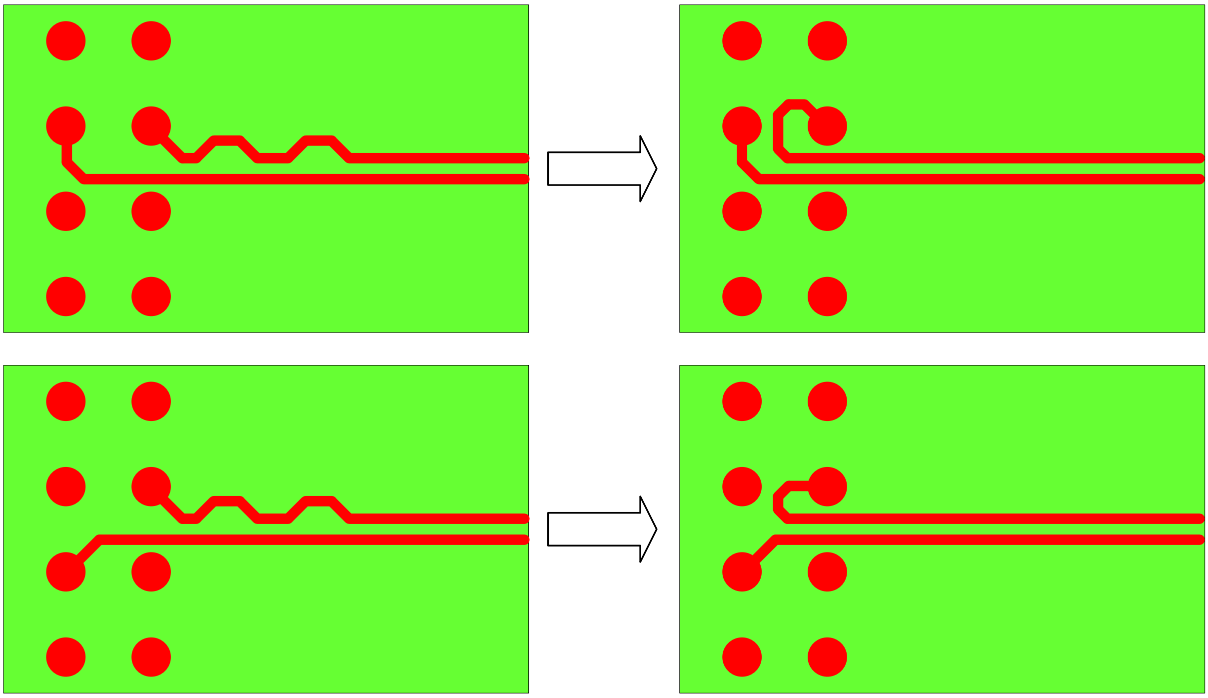


Figure 31: Preferred Breakout of Differential Pairs

6.8 Signal Return Path

An incorrect signal return path is one of the most common sources for noise coupling and EMI problems. The return path of the signal current should always be considered when routing a signal. Power rails and low speed signals take the shortest (lowest resistance) path for the return current. In contrast, the return current of high speed signals tries to follow the signal path. Differential pair signals feature a positive and negative signal trace. Even these signals require a return path which needs to be considered when routing such signals.

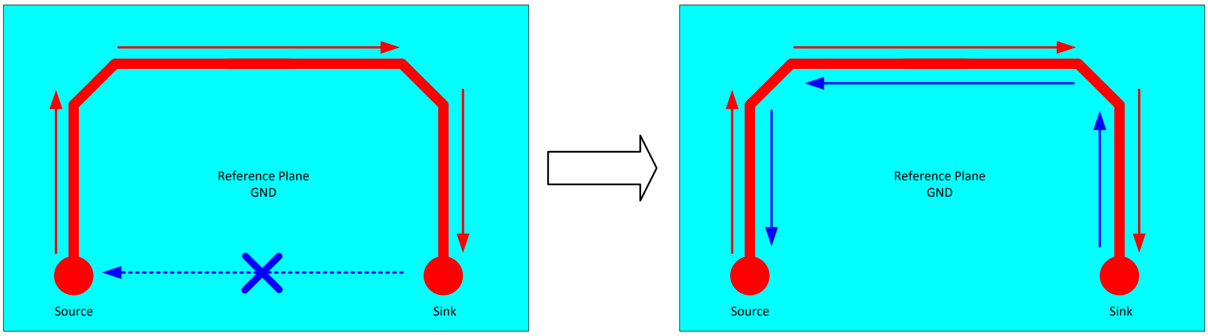


Figure 32: Return Current tries to follow the Signal Path

A signal should not be routed over a spit plane as the return path is not able to follow the signal trace. If a plane is split between a sink and source, route the signal trace around it. If the forward and return paths of a signal are separated, the area between them acts as a loop antenna.

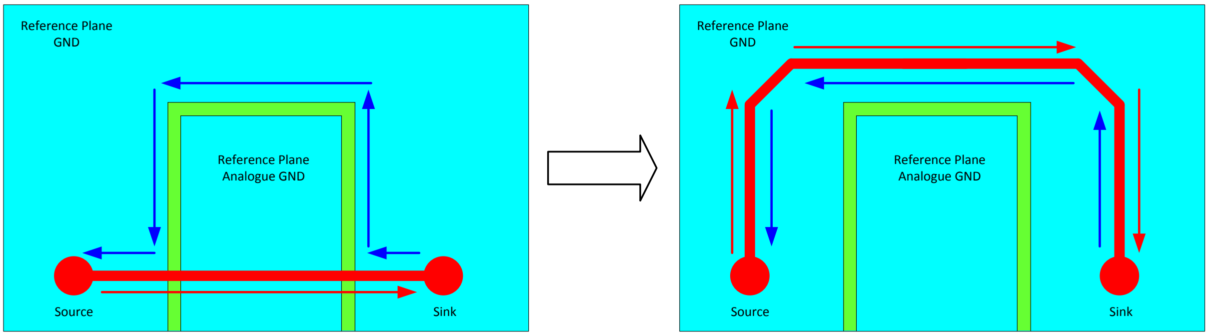


Figure 33: Avoid Routing over Split Planes

If a signal needs to be routed over two different reference planes, a stitching capacitor between the two reference planes is needed. The capacitor allows the return current to travel from one reference plane to the other. The capacitor should be placed close to the signal path in order to keep the distance between forward and return path small. A good value for the stitching capacitor is between 10nF and 100nF.

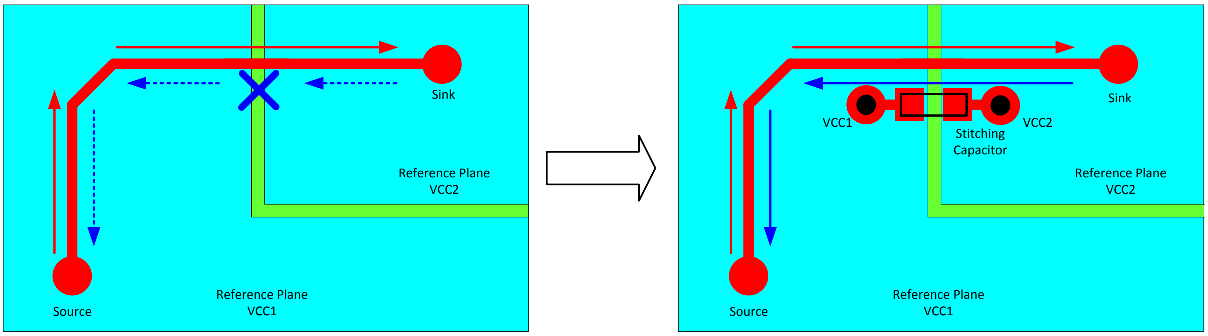


Figure 34: Stitching Capacitor needed when routed over Split Planes

Plane obstructions and plane slots should be avoided in general. Try to avoid routing signals over such obstructions. If unavoidable, stitching capacitors should be used to minimize the problems created by a separated return path.

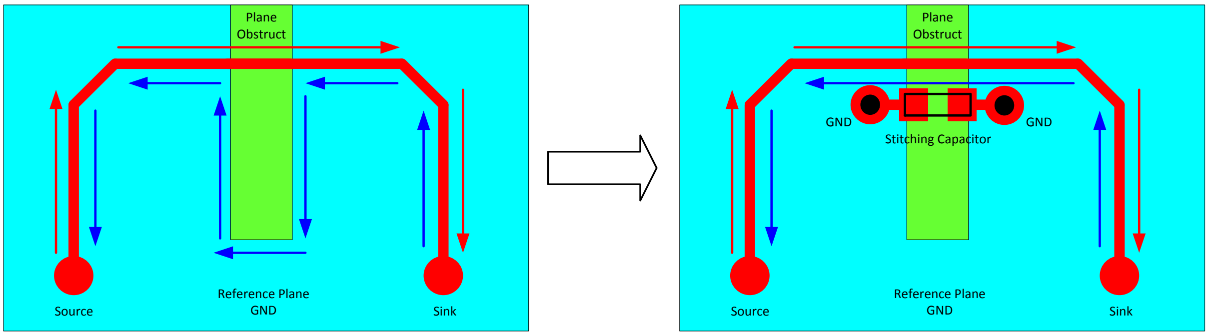


Figure 35: Stitching Capacitor needed when routing over Plane Obstructs

Voids in reference planes can result when placing vias close together. Be aware of such voids when routing high-speed signals. Try to avoid large void areas by ensuring adequate separation between vias. Sometimes, it is better to place fewer ground and power vias in order to reduce via voids.

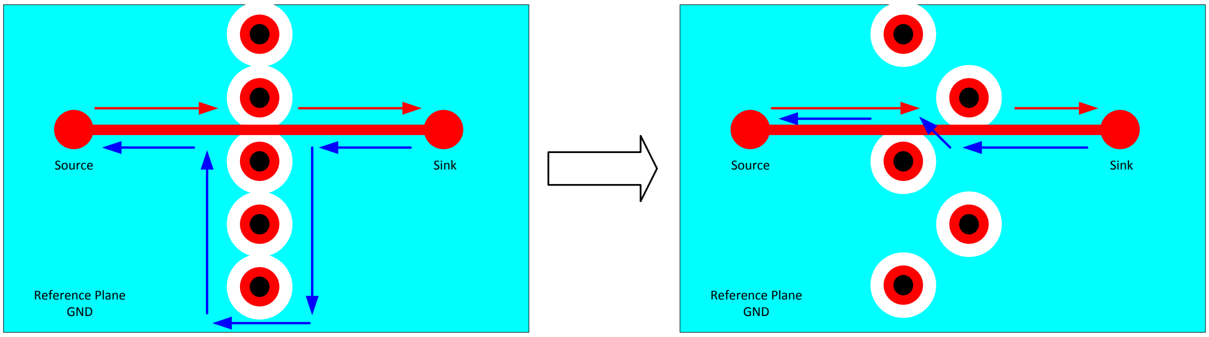


Figure 36: Avoid Via Plane Voids

The return path needs to be considered at the source and sink of a signal. The left figure below shows a bad example. As there is only one single ground via on the source side, the return current cannot travel back over the reference ground plane as intended. The return path for the current is the ground connection on the top layer instead. The problem is that the impedance of the signal trace is calculated as referenced to the ground plane and not to the ground trace on the top layer. Therefore, it is necessary to place ground vias at the source and sink side of the signal. This permits the return current to travel back on the ground plane.

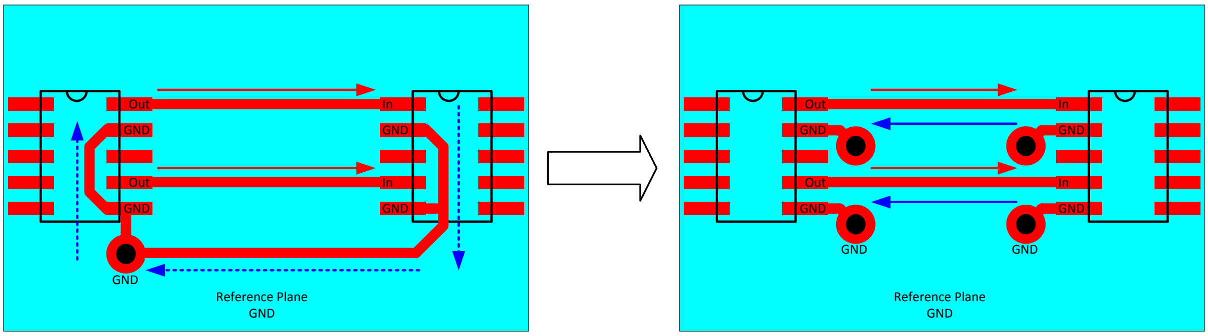


Figure 37: Consider Return Path when placing Ground Vias

When a signal trace uses a power plane as reference, the signal needs to be able to travel back over the power plane. In the source and sink, the signals are referenced to ground. In order to change the reference to the power plane, stitching capacitors at the sink and source are needed. If the sink and source are using the same power rail for their supply, the bypass capacitors can act as stitching capacitors if they are placed close to the signal entry/exit point. A good value for the stitching capacitor is between 10nF and 100nF.

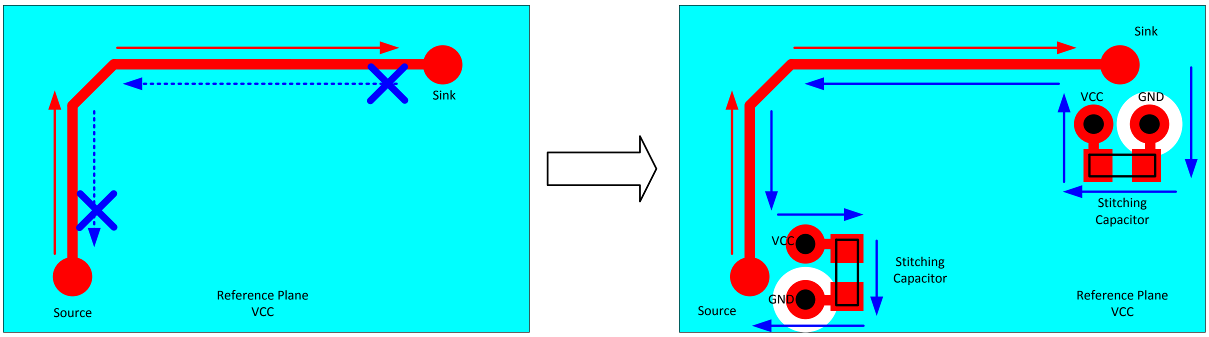


Figure 38: Add Stitching Capacitors when using Power Plane as Reference

If a signal trace switches layer and therefore, also the reference ground plane, stitching vias should to be added close to the layer change vias. This allows the return current to change ground plane. For differential signals, switching ground vias should be placed symmetrically.

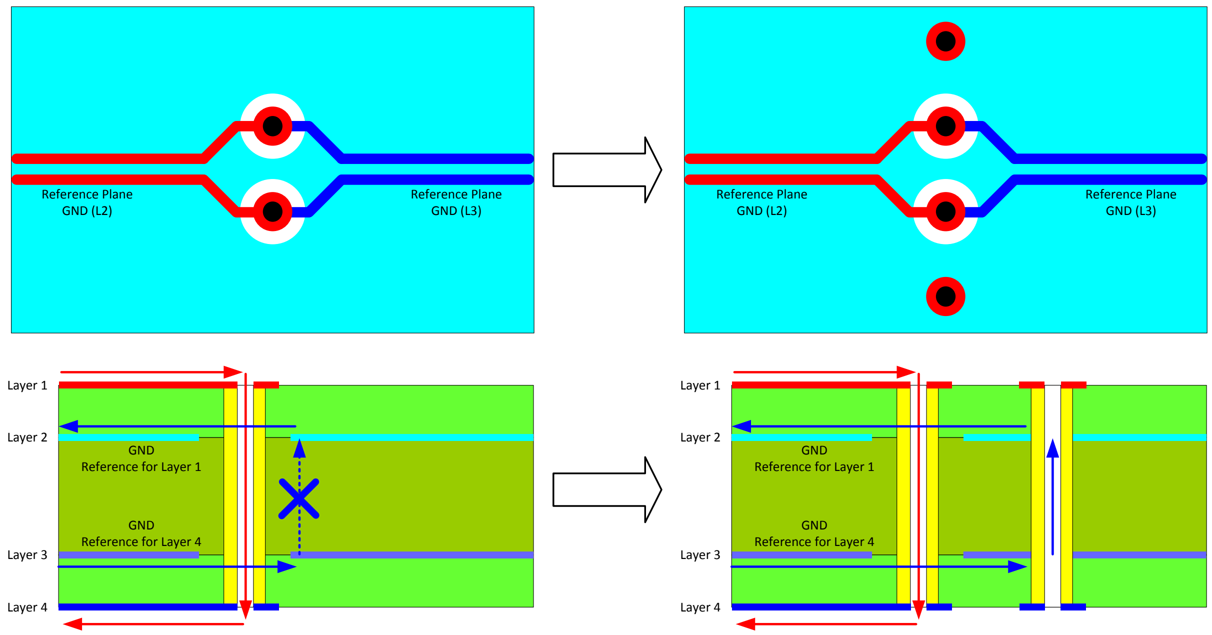


Figure 39: Place Stitching Vias when Signal changes Ground Reference

If a signal trace switches to a layer which has a different net as reference (e.g. from ground reference to power plane reference), stitching capacitors are required. This allows the return current to flow from the ground plane through the stitching capacitor to the power plane. Stitching capacitor placement and routing should be symmetrical for differential pair signals.

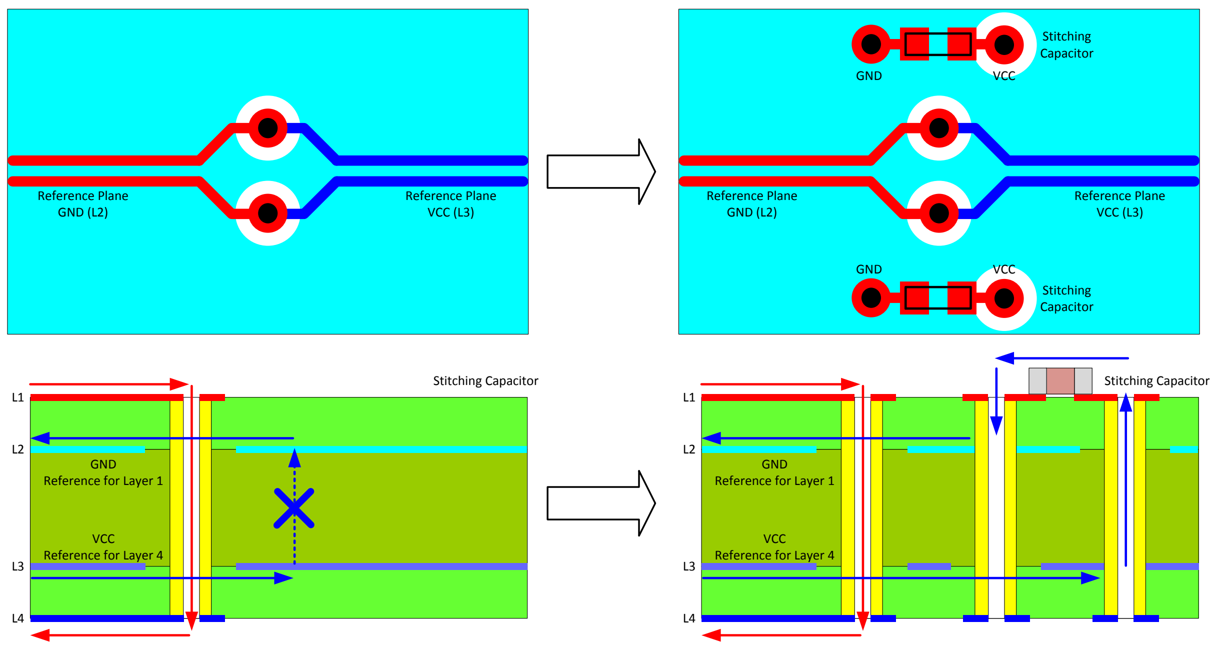


Figure 40: Place Stitching Capacitors when changing Signal Reference Plane

Avoid routing high-speed signals on the edge of reference planes or close to PCB borders. Otherwise, this can adversely affect the trace impedance.

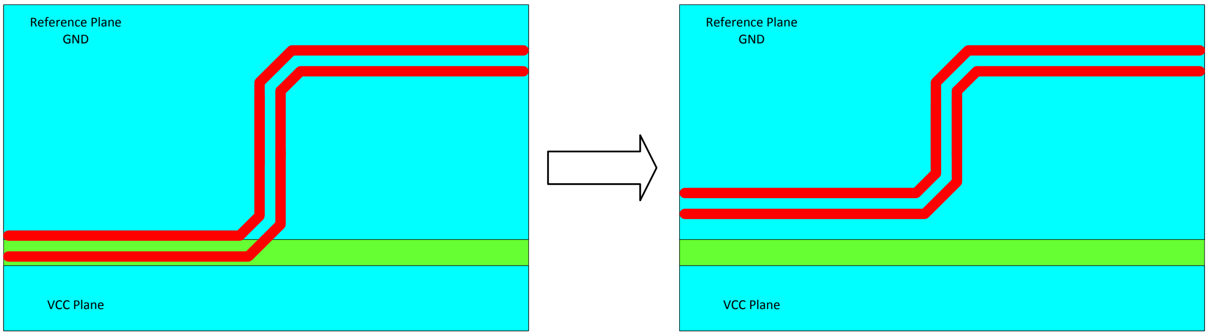


Figure 41: Do not route High Speed Signals at Plane and PCB borders

6.9 Analogue and Digital Ground

Analogue signals and circuits can be very sensitive to digital noise. There are two main coupling problems which can introduce digital noise into the analogue part. The first one is the **capacitive and inductive coupling of the signals**. This coupling can be avoided by separating the signals from each other. Special care should be taken if analogue and digital signals are routed in parallel over long distances. Increase the space between such signals as much as possible. Try to keep the analogue part away from clock signals and high current switching components (e.g. power supplies).

The second type of coupling is **conductive coupling**. The left figure shown below explains this problem. If the digital and analogue share a common return path for the power supply, the current spikes of the digital circuit can be coupled over the parasitic resistance and inductance to the analogue supply. The same coupling exists if the return path of signals is common. It is necessary to separate the return path of the digital circuits from the return path of analogue circuits.

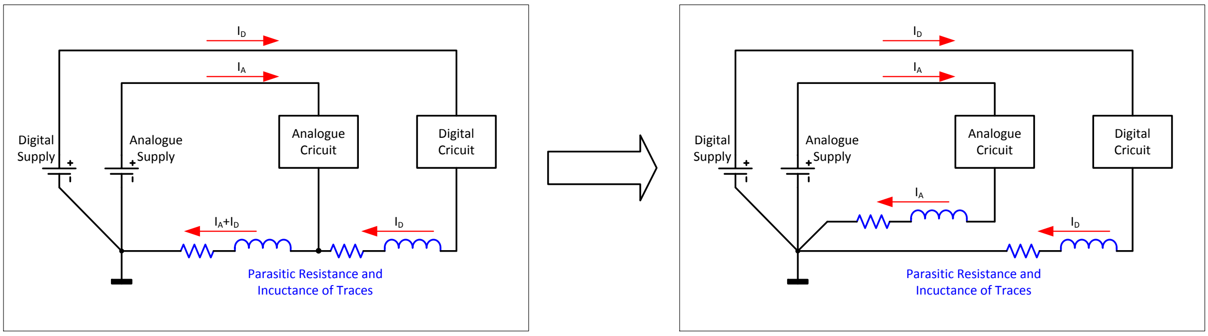


Figure 42: Separate Return Path of Analogue and Digital Supply and Signals

There are two different approaches for the separation of the digital and analogue return path (ground plane separations). The first physically divides the analogue ground planes from the digital one, and is referred to as the “Split Plane Approach”. The second divides the grounds virtually, and is similarly referred to as the “Virtual Split Plane Approach”. Both approaches have their advantages which make it difficult to judge which one amongst them is better.

6.9.1.1 Split Plane Approach

A lot of reference schematics for mixed signal integrated circuits (e.g. ADC) propose a split ground approach. It makes it easy in the schematic to show which components and pins should be connected to digital ground and which ones to the analogue ground. Such schematics can be routed by placing two different ground planes as reference. The two planes need to be placed carefully. The analogue ground should only be placed under analogue pins and components. This requires careful placement of the components.

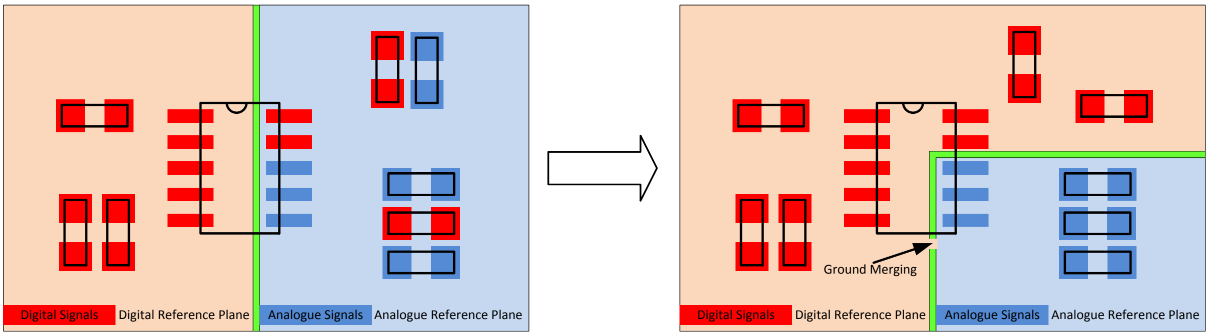


Figure 43: Power Plane Splitting need to be placed carefully

Mixed signal circuits need to have the analogue and digital ground connected together at a single location. In reference schematics it is often recommend to place ferrite beads or zero ohm resistors between the two nets. The merging of the digital and analogue ground should be placed close to the integrated circuit which features both analogue and digital signals.

In a mixed-signal design with split planes it is important that no digital signal is routed over an analogue ground plane while no analogue signal is routed over the digital ground plane. The two domains need to be completely separated.

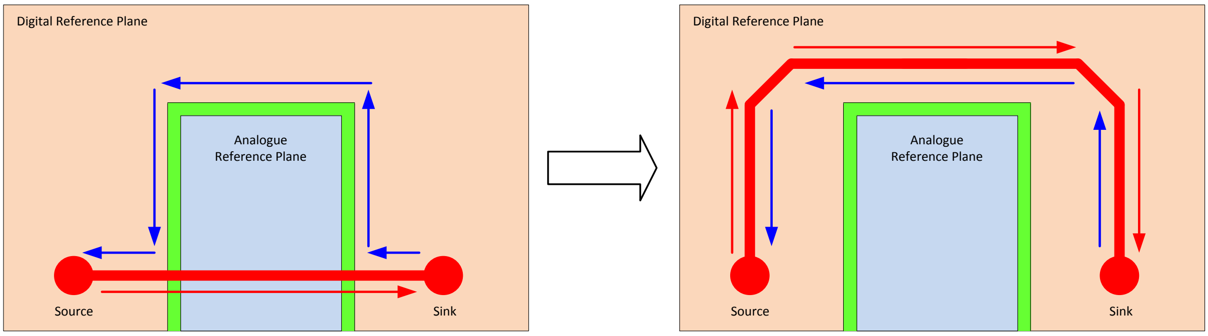


Figure 44: No Digital Signals are allowed to cross the Analogue Ground Plane

One of the advantages of the split plane approach is that it is clear in the schematic which ground connections are digital and which ones are analogue. Also, the separation between the two domains is clearly visible in the layout. Schematic and layout can be less confusing if more than one engineer is working on the project.

6.9.1.2 Virtual Split Approach

The virtual split approach does not split the analogue and digital ground in the schematic diagram. In the layout, the two ground domains are not electrically split. The trick is to implement the layout as if there is an imaginary separation between the analogue and digital ground. Some CAD tools allow drawing a help line on an unused mechanical layer. The components need to be placed carefully on the correct side of the virtual split planes.

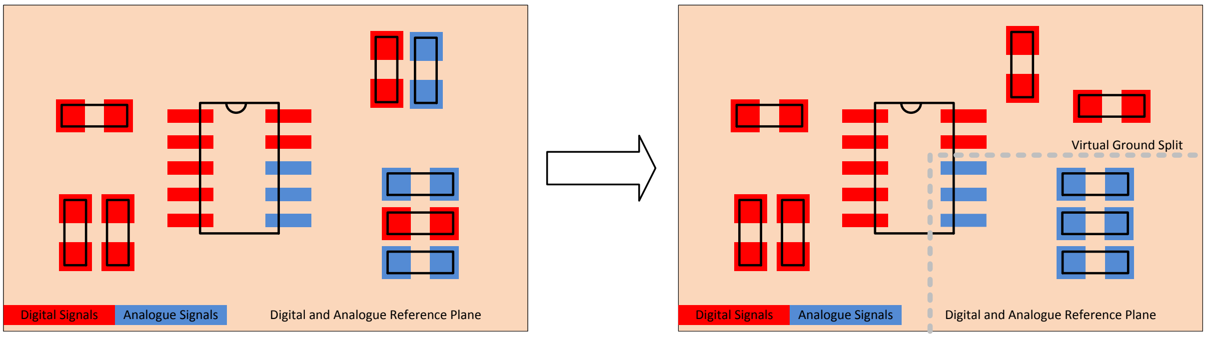


Figure 45: Careful Component Placement needed even with Virtual Plane Splitting

The virtual line between the two ground domains needs to be respected during the routing. No digital or analogue signal trace is permitted to cross the virtual split line. The virtual split line should not be placed using a very complicated shape as there are no plane obstructs to keep the analogue and digital return current separated.

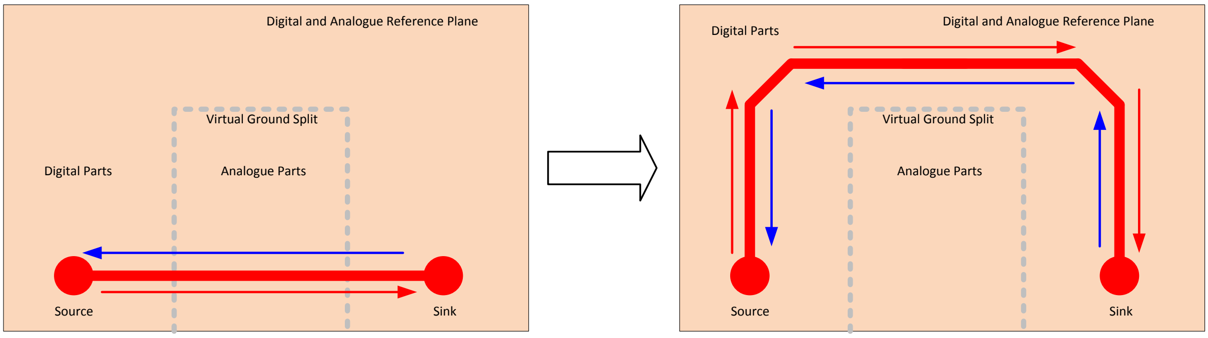


Figure 46: No Digital Signals are allowed to cross the Virtual Analogue Ground

Routing can be more challenging with the virtual split plane approach as it is easy to make an improper separation of the two domains which will not be picked up by the DRC. Special care has to be taken that the analogue and digital ground parts are correctly split. If the layout is correctly implemented with this approach, a better solution can be achieved than with the physical split plane approach.

7 Layout Requirements of Interfaces

Depending on the type of interface, there are different layout requirements available. This section provides an overview of the major interfaces that can be found on an Apalis or Colibri module. If the information is missing for an interface, check the according module datasheet or contact the Toradex support team.

The differential pair signals normally distinguish between two different length matching requirements. The first requirement is the maximum intra-pair skew. This is the maximum allowed length differences between the positive and negative signal of the pairs. As described in section 6.7, not only should the overall length be matched, but also the length within a section of the signal should be corrected. It is important that the positive and negative signal components are propagated synchronously. Only if these signals are synchronous, their fields are compensated and the electromagnetic radiation is reduced.

The second length matching requirement is the maximum allowed skew between the clock and signal pairs or between different pairs of the same interface. Some of the interfaces (e.g. PCIe, SATA, and USB3.0) are recovering the clock signal out of the data signals. Therefore, the matching between the clock and data signals can be quite relaxed (e.g. 240mm). Do not try to overmatch such interfaces since it is really not required and the additional meander just introduces other signal quality problems. On the other hand there are interfaces which do not have an embedded clock signal (e.g. LVDS LCD interface). Please route these signals very carefully. The length matching between the clock and data signals needs to be met (e.g. 0.5mm).

Vias introduce a major discontinuity of the impedance and can create signal stubs. Therefore, the amount of vias should be kept as low as possible. Of course, some vias cannot be avoided. Therefore, some of the interfaces have a budget of maximum amount of vias from in the complete connection. The following sections show the maximum allowed vias on the carrier board, not in the complete connection.

7.1 PCI Express

The PCIe interface supports the Polarity Inversion. This means the positive and negative signal pins can be inverted in order to simplify the layout by avoiding crossing of the signals. Some PCIe devices support additional Lane Reversal for multi-lane interfaces. Since the standard interfaces of Apalis provide only a single lane PCIe interface, the Lane Reversal feature is not supported in the Apalis specifications. Some Apalis modules provide additional multi-lane PCIe interface in the type-specific area. For such modules, Lane Reversal is only applicable if the PCIe device is supporting it.

Parameter	Requirement
Max Frequency	Gen1: 1.25GHz (2.5GT/s) Gen2: 2.5GHz (5GT/s) Gen3: 4GHz (8GT/s)
Configuration / Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	90Ω ±15% differential; 50Ω ±15% single ended
Max intra-pair skew	<1ps ≈150µm
Max trace length skew between clock and data pairs	<1.6ns ≈240mm
Max trace length on carrier board (device-down)	<300mm
Max trace length on carrier board (PCIe slot connector, PCIe Mini card)	<200mm
Minimum pair to pair spacing	>500µm
AC coupling capacitors	100nF ±20%, discrete 0402 package preferable
Maximum allowed via	2 vias for TX traces 4 vias for RX traces (device-down) 2 vias for RX traces (PCIe slot connector, PCIe Mini card)

Table 5: PCIe Layout Requirements

7.2 SATA

The SATA interface does not support the Polarity Inversion. This means the positive and negative signal pins are not allowed to be swapped for layout simplification.

Parameter	Requirement
Max Frequency	SATA I: 750MHz (1.5GT/s) SATA II: 1.5GHz (3GT/s) SATA III: 3GHz (6GT/s)
Configuration / Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	90Ω ±15% differential; 55Ω ±15% single ended
Max intra-pair skew	<1ps ≈150µm
Max trace length skew between data pairs	<0.8ns ≈120mm
Max trace length on carrier board	<150mm
Minimum pair to pair spacing	>500µm
AC coupling capacitors	No coupling capacitors needed on the carrier board
Maximum allowed via	2 vias for RX and TX traces

Table 6: SATA Layout Requirements

7.3 Ethernet

The Ethernet MDI signals are analogue differential pair signals which need to be routed carefully. Try to keep the MDI signals as short as possible and keep them away from digital signals. Try to avoid any stubs on these signals.

Parameter	Requirement
Max Frequency	10Base-T: 10MHz (10Mbit/s) 100Base-TX: 31.25MHz (100Mbit/s) 1000Base-T: 62.5MHz (1Gbit/s)
Configuration / Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	95Ω ±15% differential; 55Ω ±15% single ended
Max intra-pair skew	<1.6ps ≈250µm
Max trace length skew between data pairs	<330ps ≈50mm
Max trace length on carrier board between module connector and magnetics	<~100mm, keep it as short as possible
Max trace length on carrier board between magnetics and Ethernet Jack (discrete magnetics)	Place magnetics as close as possible to the Ethernet connector in order to reduce EMC and ESD problems. Separate the reference ground under the traces between magnetics and Ethernet connector. Keep a minimum distance of 2mm between this ground and the common ground.
Minimum pair to pair spacing	>450µm
Minimum spacing between MDI signals and other high speed signals	>7.5mm
Minimum spacing between MDI signals and low speed signals	>2.5mm
Maximum allowed via	2 vias for all MDI traces

Table 7: Ethernet Layout Requirements

If discrete magnetics are used instead of a RJ-45 Ethernet jack with integrated magnetics, special care has to be taken to route the signals between the magnetics and the jack. These signals are required to be of high voltage and isolated from other signals. It is therefore necessary to place a dedicated ground plane under these signals which has a minimum separation of 2mm from every other signal and plane. Additionally, a separate shield ground for the LAN device is needed. Try to place the magnetics as close as possible to the Ethernet jack. This reduces the length of the signal traces between the magnetics and jack.

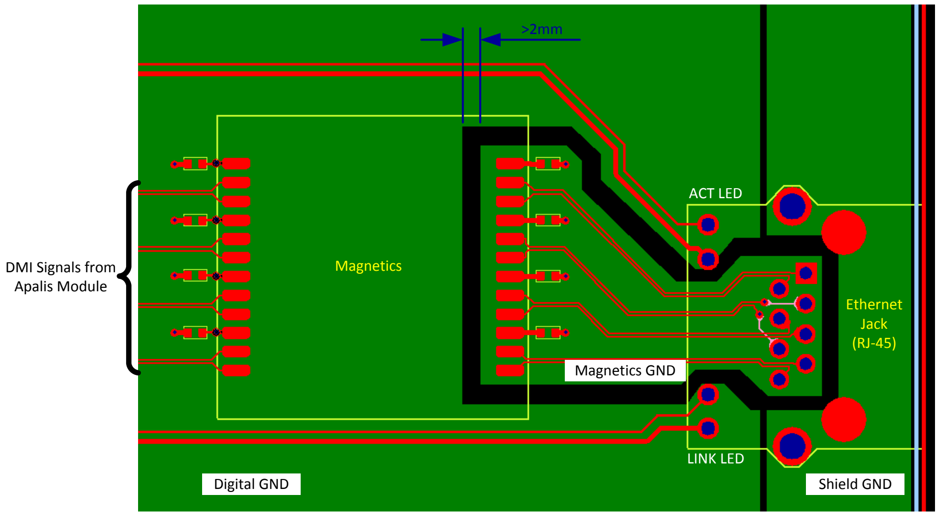


Figure 47: Separation of Magnetics Ground

7.4 USB

The layout requirement for USB interfaces depends on the version that needs to be supported. Up to USB 2.0, the interface was consisting of a single bidirectional data signal pair. The USB 3.0 standard introduces two additional data signal pairs for the Super Speed link. These two pairs are running at 5 Gbit/s and are fully compliant to the PCI Express Base Specification, Revision 2.0. Super Speed signals support the Polarity Inversion. This means the positive and negative signal pins can be inverted in order to simplify the layout by avoiding crossing of the signals. But it is not allowed to swap the receiving signals with the transmitting ones. The USB 2.0 data signals do not support Polarity Inversion, D+ and D- cannot be swapped.

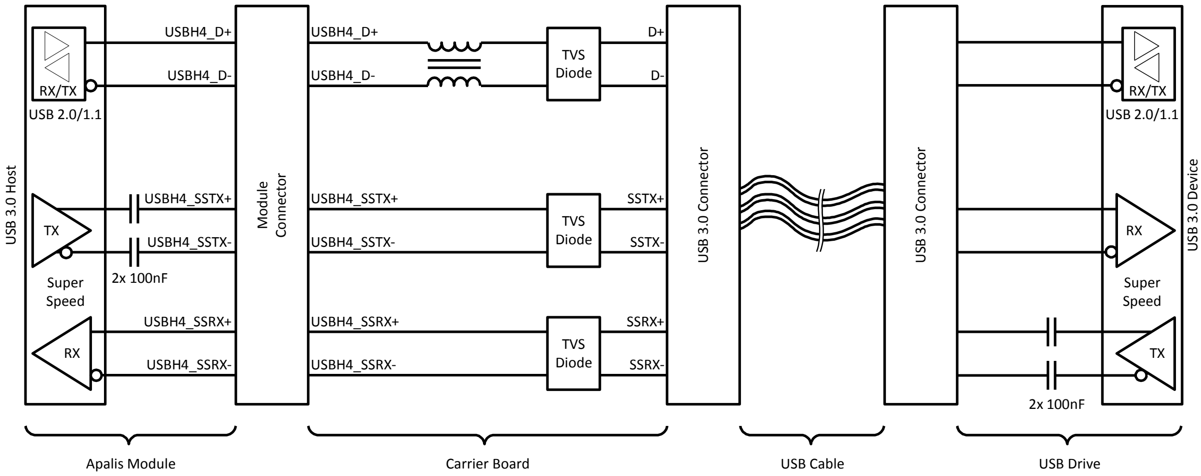


Figure 48: USB 3.0 Block Diagram

7.4.1.1 USB 2.0 Signals

Parameter	Requirement
Max Frequency	Low Speed: 750kHz (1.5Mbit/s) Full Speed: 6MHz (12Mbit/s) High Speed 240MHz (480Mbit/s)
Configuration / Device Organization	1 load (10pF High Speed, 150pF Full Speed, 600pF Low Speed)
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	90Ω ±15% differential; 50Ω ±15% single ended
Max intra-pair skew	<7.5ps ≈1.1mm
Max trace length on carrier board (USB connector)	<200mm
AC coupling capacitors	No coupling capacitors allowed
Maximum allowed via	A minimum amount of vias should be used.

Table 8: USB2.0 Layout Requirements

7.4.1.2 USB 3.0 Signals

Parameter	Requirement
Max Frequency	Super Speed: 2.5GHz (5GT/s)
Configuration / Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	90Ω ±15% differential; 50Ω ±15% single ended
Max intra-pair skew	<1ps ≈150µm
Max trace length skew between RX and TX data pairs	<1.6ns ≈240mm
Max trace length on carrier board (USB3.0 connector)	<200mm
Minimum pair to pair spacing	<500µm
AC coupling capacitors	100nF ±20%, discrete 0402 package preferable
Maximum allowed via	2 vias for TX traces 4 vias for RX traces (device-down) 2 vias for RX traces (USB3.0 connector)

Table 9: USB3.0 Layout Requirements

7.5 Parallel RGB LCD Interface

The layout requirements depend on the pixel clock and therefore on the required display resolution. The requirements below can be greatly relaxed if lower resolutions such as VGA 640x480 are used. The maximum length restrictions are defined due to electromagnetic radiation problems associated with the parallel interface. From the timing perspective, the trace length of the interface is not limited.

Parameter	Requirement
Max Frequency	Depending on maximum resolution of the module (e.g. 162MHz for VESA 1600x1200@60Hz). Check the datasheet of the Apalis module.
Configuration / Device Organization	1 load (multiple load possible for lower resolutions)
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	50Ω ±15% single ended
Max skew between data signal and clock	<100ps ≈15mm, depends on pixel clock, requirement can be relaxed for lower resolution displays
Max trace length (RGB displays, including flex cable length)	<100mm (long distances only recommended for low resolution displays)
Max trace length (converter on carrier board)	<50mm

Table 10: Parallel RGB LCD Interface Layout Requirements

7.6 LVDS LCD Interface

The LVDS LCD interface does not have the clock embedded in the data signals. Therefore, a proper length matching between the clock pair and the data signals is essential and depends on the display resolution.

Parameter	Requirement
Max Frequency	Depending on maximum resolution of the module. The maximum frequency is 7 times higher than the pixel clock in single channel mode.
Configuration / Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	100Ω ±15% differential; 55Ω ±15% single ended
Max intra-pair skew	<1ps ≈150µm
Max trace length skew between clock and data pairs	<3.5ps ≈500µm, depends on LVDS frequency since clock is not embedded, can be relaxed for lower resolutions
Max trace length on carrier board and display cable	<500mm
Minimum pair to pair spacing	>2x intra-pair spacing
Maximum allowed via	Minimize the number of via in LVDS traces

Table 11: LVDS Layout Requirements

7.7 HDMI/DVI

Since the external HDMI or DVI cables can be quite long, it is desirable to have a certain skew between the different data pairs. This can help to reduce the electromagnetic emissions.

Parameter	Requirement
Max Frequency	Version 1.0-1.2a: 825MHz (165 MHz pixel clock) Version 1.3-1.4: 1.65GHz (340 MHz pixel clock)
Configuration / Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	90Ω ±15% differential; 50Ω ±15% single ended
Max intra-pair skew	<5ps ≈750µm
Max trace length skew between clock and data pairs	<150ps ≈22mm
Max trace length on carrier board	<250mm
Minimum pair to pair spacing	>500µm
Maximum allowed via	Minimize the number of via in TDMS traces

Table 12: TDMS Signal Layout Requirements

7.8 Analogue VGA

Parameter	Requirement
Max Frequency	Depending on resolution, up to 388MHz for 2048x1536@85Hz
Configuration / Device Organization	1 load
Reference Plane	AGND or GND (if GND is used, do not cross with digital signals the reference plane)
Trace Impedance	50Ω ±15% in section B 75Ω ±15% in section C (if 75Ω is not achievable, use 50Ω and keep traces short)
Max skew between analogue RGB color signal	Try to match the trace length as close as possible
Max trace length on carrier board section B	<200mm (longer distances achievable when layout is carefully done)
Max trace length on carrier board section C	<15mm (try to keep section C as short as possible)
Minimum spacing between color signals	>350µm
Minimum spacing between color signals and other signals	>500µm from low speed signals >1.25mm from clock signals and high current power traces >6mm from areas with high switching currents (e.g. voltage regulators)
Maximum allowed via	Minimize the number of via in the analogue RGB traces

Table 13: Analogue VGA Signal Layout Requirements

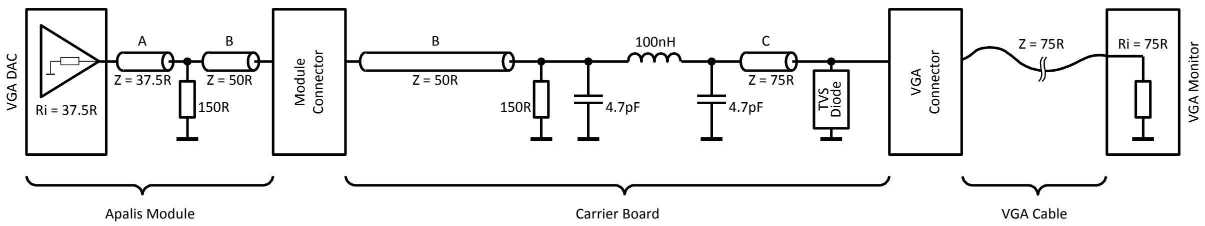


Figure 49: VGA Signal Trace Impedance Sections

7.9 Parallel Camera Interface

The requirements for the layout depends on the pixel clock and therefore on the required camera resolution and frame rate. The requirements below can be relaxed if lower resolutions such as VGA 640x480@30Hz are used. The maximum length restriction is given due to EMC compliance problems which otherwise may occur. From the timing perspective, the trace length of the interface is not limited.

Parameter	Requirement
Max Frequency	Depending on maximum resolution of the camera and the frame rate.
Configuration / Device Organization	1 source
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	50Ω ±15% single ended
Max skew between data signal and clock	<100ps ≈15mm, depends on pixel clock, requirement can be relaxed for lower resolution displays
Max trace length	<50mm

Table 14: Parallel Camera Interface Layout Requirements

7.10 SD/MMC/SDIO

Parameter	Requirement
Max Frequency	208MHz (Ultra-High Speed SD, SDR104) 52MHz (MMCplus, eMMC)
Configuration / Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	50Ω ±15% single ended
Max skew between data signal and clock 52MHz MMC	<100ps ≈15mm
Max skew between data signal and clock SDR104	<20ps ≈3mm
Max trace length	<100mm

Table 15: SD/MMC/SDIO Layout Requirements

7.11 I²C

The I²C does not need to be routed as differential pair, but it is recommended not to separate data and clock line too much. It is not required to rout the bus as daisy chain since the stub length is not a problem. The maximum trace length is limited due to the capacity load of the traces and attached bus devices. Therefore, try to keep traces as short as possible by using a star topology.

Parameter	Requirement
Max Frequency	100kHz (Standard Mode) 400kHz (Fast Mode)
Configuration / Device Organization	Multiple load, 400pF (Standard Mode), 100pF (Fast Mode)
Reference Plane	GND or PWR (No stitching capacitors required when PWR is used as reference)
Trace Impedance	50Ω ±15% single ended
Max trace length (Standard Mode)	<450mm (is mainly limited by the capacitive load)
Max trace length (Fast Mode)	<200mm (is mainly limited by the capacitive load)

Table 16: I²C Layout Requirements

7.12 Display Serial Interface (MIPI/DSI with D-PHY)

Parameter	Requirement
Max Frequency	500MHz (1GT/S per data lane)
Configuration/Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	90Ω ±15% differential; 50Ω ±15% single ended
Max Intra-pair Skew	<1ps ≈150µm
Max Trace Length Skew between clock and data lanes	<10ps ≈1.5mm
Max Trace Length from Module Connector	200mm
Maximum allowed via	Minimize the number of via in each lane

Table 17: DSI Layout Requirements

7.13 Camera Serial Interface (MIPI/CSI-2 with D-PHY)

Parameter	Requirement
Max Frequency	500MHz (1GT/S per data lane)
Configuration/Device Organization	1 load
Reference Plane	GND or PWR (if PWR, add 10nF stitching capacitors between PWR and GND on both sides of the connection for the return current)
Trace Impedance	90Ω ±15% differential; 50Ω ±15% single ended
Max Intra-pair Skew	<1ps ≈150µm
Max Trace Length Skew between clock and data lanes	<10ps ≈1.5mm
Max Trace Length from Module Connector	200mm
Maximum allowed via	Minimize the number of via in each lane

Table 18: CSI-2 Layout Requirements