Board Design Guidelines for LVDS Systems

This white paper explains the basic PCB layout guidelines for designing low-voltage differential signaling (LVDS) boards using Altera® FPGAs.

Introduction

LVDS is a high-speed, low-voltage, low-power, and low-noise general-purpose I/O interface standard. The low-voltage swing and differential current mode outputs significantly reduce electromagnetic interference (EMI). These outputs have fast edge rates that cause signal paths to act as transmission lines. Therefore, ultra-high-speed board design and differential signal theory knowledge is especially useful for designing a board containing Altera FPGAs that integrate LVDS. In addition, a number of factors, such as differential traces, impedance matching, crosstalk, and EMI, have to be considered while designing an LVDS board.

Differential Traces

LVDS utilizes a differential transmission scheme, which means that every LVDS signal uses two lines. The voltage difference between these two lines defines the value of the LVDS signal. For successful transmission of LVDS signals over differential traces, the following guidelines should be followed while laying out the board.

- To ensure minimal reflections and maintain the receiver's common mode noise rejection, run the differential traces as closely as possible after they leave the driving IC. Also, to avoid discontinuities in the differential impedance, the distance between the differential LVDS signals should remain constant over the entire length of the traces.
- To minimize skew, the electrical lengths between the differential LVDS traces should be the same. Arrival of one of the signals before the other creates a phase difference between the signal pair, which impairs the system performance by reducing the available receiver skew margin (RSKM).
- Minimize the number of vias or other discontinuities on the signal path.
- Any parasitic loading, such as capacitance, must be present in equal amounts to each line of the differential pair.
- To avoid signal discontinuities, arcs or 45° traces are recommended instead of 90° turns.

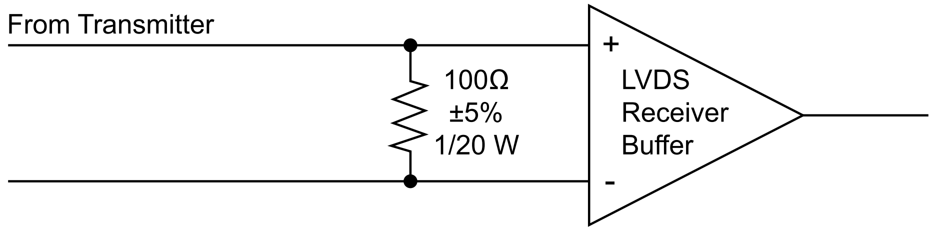
Impedance Matching

Due to the high speed of LVDS, impedance matching is very important, even for very short runs. Any discontinuities in the differential LVDS traces will cause signal reflections, thereby degrading the signal quality. These discontinuities also increase the common mode noise and will be radiated as EMI. The LVDS outputs, being current mode outputs, need a termination resistor to close the loop and will not work without the resistor termination. The value of this

101 Innovation Drive
San Jose, CA 95134
www.altera.com

termination resistor (RT) is chosen to match the differential impedance of the transmission line and can range from 90Ω to 110Ω (typically 100Ω). Figure 1 shows the correct usage of the termination resistor.

Figure 1. LVDS Termination Scheme



The following guidelines should be used when selecting the termination resistor for an LVDS channel.

- Place the termination resistor at the far end of the differential interconnect from the transmitter. A single 100Ω resistor is sufficient.
- Use surface-mount thick-film 0603- or 0805-size chip resistors.
- Install the termination resistor within 7 mm of the receiver, as close to the receiver as possible.

Crosstalk Between LVDS and Single-Ended Signals

To reduce crosstalk between LVDS and single-ended signals such as LVTTL, SSTL-3, SSTL-2, and similar standards, the differential LVDS signals must be isolated from single-ended signals. If the LVDS and single-ended signals are not placed sufficiently apart from one another, the single-ended signals may cause some interference on the differential pair. The LVDS signal that runs closest to the single-ended signal trace will be affected more than the farther one, creating a difference that will not be rejected by the LVDS receiver as common mode noise. This interference is unlikely to cause the LVDS receiver to falsely trigger; however, it will degrade the signal quality of the LVDS signal, thereby reducing the noise margin. On the same PCB layer, the single-ended signals should be placed at least 12 mm from the LVDS signals to avoid crosstalk effects. The VCC and ground planes can also be used to isolate the LVDS signal layers from the single-ended signal layers. Figure 2 shows the shielding of the LVDS layers from the single-ended layers using the power planes.

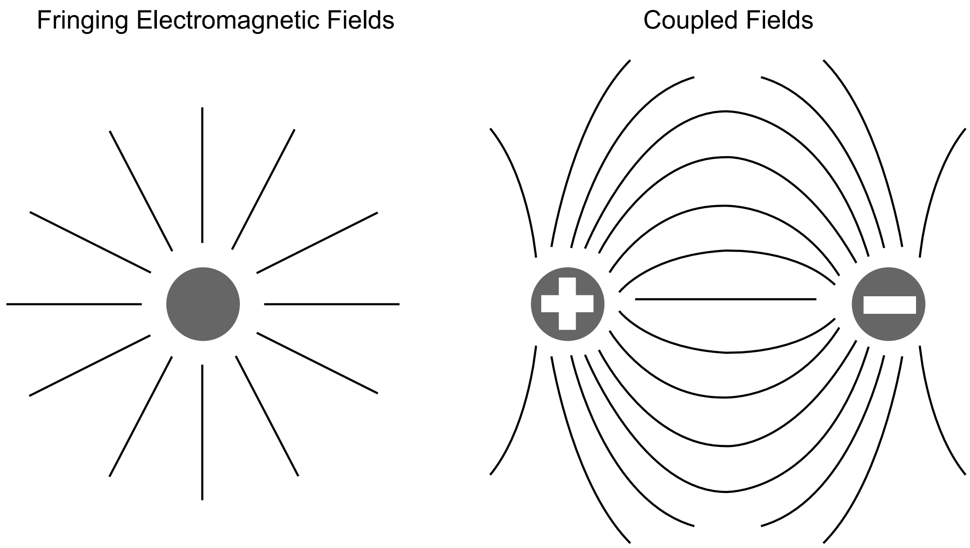
Figure 2. Power Planes

Electromagnetic Interference

Electromagnetic radiation is usually a cause for concern for designers because this radiation can propagate through transverse electromagnetic (TEM) waves. These waves can escape through shielding, causing a system to fail electromagnetic compliance (EMC) tests. With single-ended transmission such as CMOS or TTL, almost all of the field lines are free to radiate away from the conductor. Some of these field lines can travel as TEM waves, which may escape the system and thereby cause EMC problems.

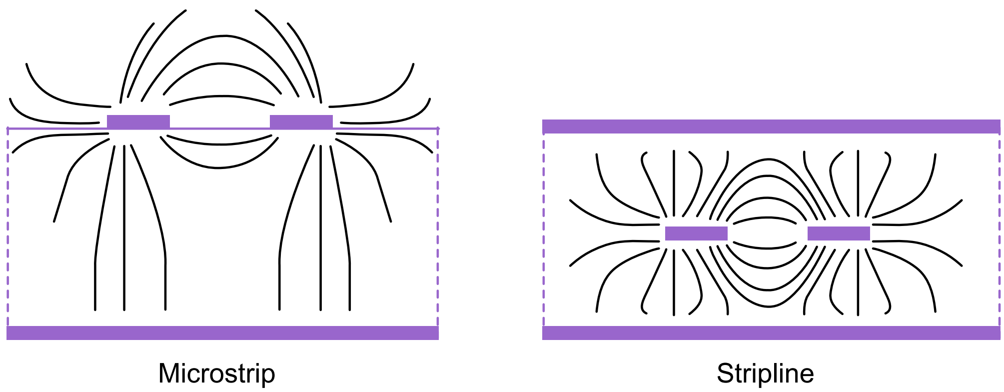
For LVDS differential signals, field lines tend to cancel each other out, and the electric fields tend to couple. These coupled fields are tied up with each other and thus are not allowed to escape. Only a few fringing fields escape out of this coupling. Therefore, LVDS, being a differential transmission system, generates less EMI compared to CMOS or TTL signals. Figure 3 shows the electromagnetic field effects in single-ended traces and differential pairs.

Figure 3. Electromagnetic Field Effects



LVDS signals can be routed on the PCB microstrip (external layers) and stripline (middle layers). Figure 4 shows the electromagnetic field radiation for LVDS stripline and microstrip traces.

Figure 4. Microstrip and Stripline Differential Pair Dimensions

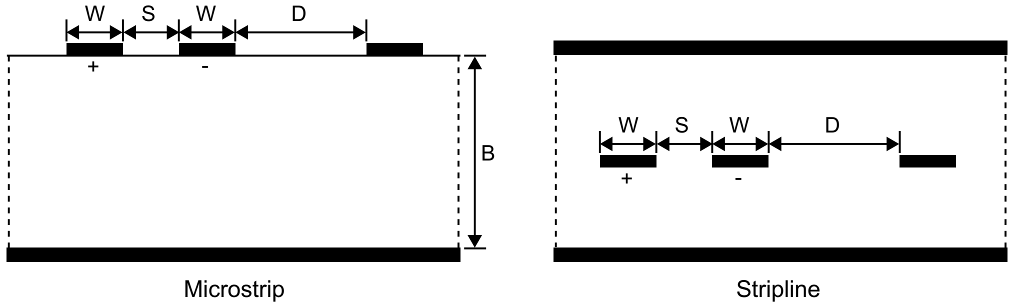


For microstrip, the ground plane below couples additional field lines, thereby tying up more field lines and reducing the EMI effects. Stripline couples all of the field lines, thereby reducing EMI significantly, but it also has the following penalties:

- Considerably higher (typically one and a half times) propagation times than that of microstrip
- Needs additional vias
- Needs more layers
- Difficulty in achieving 100Ω differential impedance accurately

In order to have maximum coupling of the magnetic field lines, the space between two conductors of a differential pair should be kept to a minimum. Figure 5 shows the dimensions of a stripline and microstrip pair.

Figure 5. Microstrip and Stripline Differential Pair Dimensions



For better coupling within a differential pair, make $S < 2W$, $S < B$, and $D = 2S$ where:

- W = width of a single trace in a differential pair
- S = space between two traces of a differential pair
- D = space between two adjacent differential pairs
- B = thickness of the board

For good coupling between two conductors of a differential pair, the following rules should be followed:

- Space between the conductors should not be more than twice the width ($S < 2W$)
- Thickness of the board should be more than the space between the conductors ($B > S$)
- Space between two adjacent differential pairs should be greater than or equal to twice the space between the two individual conductors. ($D > 2S$)

General PCB Guidelines

This section lists general PCB layout and supply voltage guidelines.

- The commonly used FR-4 material works well for low frequency (500 to 600 MHz) applications. G-TEK or Teflon can be considered for high-speed designs.
- Estimate the number, value, and type of decoupling capacitors required to develop an efficient PCB decoupling strategy during the early design phase, without going through extensive pre-layout simulations. Altera's Power Delivery Network (PDN) tool provides these critical pieces of information.

For further information about the PDN tool that targets your FPGA, refer to the Power Distribution Network Design Tool webpage.

- When using LVDS devices, all the `VCC_CKLK` and `VCC_CKOUT` pins should be bypassed with a 0.1-, 0.01-, and 0.001 µF mica, ceramic or polystyrene 0805-size surface-mount chip capacitors connected in parallel. These capacitors should be placed immediately underneath the pins. In addition to these capacitors, another 2.7 µF capacitor should be placed close to the pin.
- Keep the LVDS drivers and the receiver as close to any connectors as possible.
- The physical length of each trace between the transmitter outputs and the connector should be matched to within 5 mm of each other to reduce data skew.
- Isolate LVDS signals from TTL signals to reduce crosstalk (preferably on different layers).
- Separate LVDS ground and supply planes.
- Always use high-impedance, low-capacitance scope probes with a wide bandwidth scope.
- Keep stub lengths as short as possible.
- Multiple vias should be used to connect to power and ground planes.

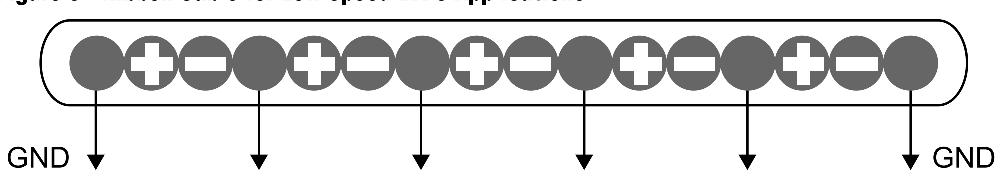
LVDS Cables

Cables can be used to transfer the LVDS signals from one board to another. However, due to the specific impedance matching and low skew requirements, typical cables may not be suitable for LVDS. Use the following guidelines to select cables for LVDS applications:

- The cables must fulfill the “matched impedance” requirement of LVDS.
- Cables should have very low skew.
- The conductor pairs must be balanced (i.e., both of the conductors should incur the same delay while going through the cable).

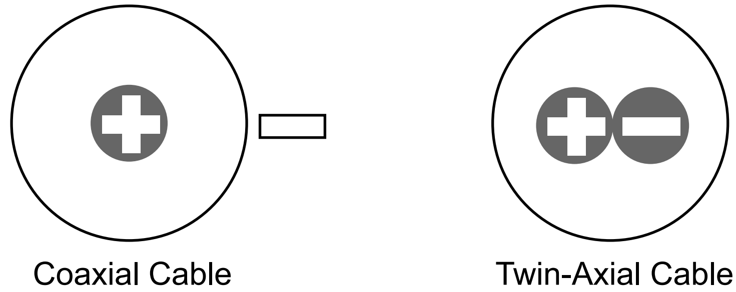
For low speed and very short runs, ribbon cables can be used. For longer runs and high speed, use twisted-pair cables (balanced twisted-pair cables work well in this application). If you use ribbon cables, the signal pairs must be separated by the ground wires, and the edge conductors of the ribbon cable should not be used to carry signals. Figure 6 shows a ribbon cable used for LVDS application.

Figure 6. Ribbon Cable for Low Speed LVDS Applications



Twin-axial cables can also be used for LVDS applications, as they are far more balanced then coaxial cables. In contrast to coaxial cables, twin-axial cables generate far less EMI due to the field cancellation effects. The coaxial and twin-axial cables are shown in Figure 7.

Figure 7. Coaxial and Twin-Axial Cables

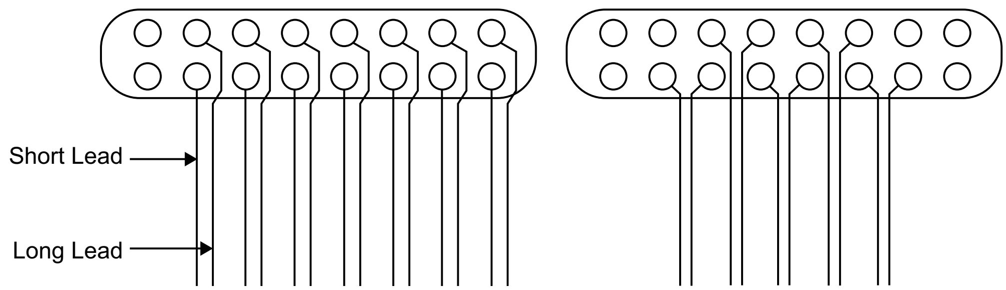


For optimum performance, use twisted-pair cables, because the LVDS receiver rejects whatever small common-mode electromagnetic radiation these cables pick up. For small distances (approximately 0.5 m), the CAT3 balanced twisted-pair cables work well. For distances greater than 0.5 m and data rates greater than 500 MHz, use CAT5 balanced cables.

LVDS Connectors

Connectors can be used to connect the LVDS signals from one board to another. Figure 8 shows good and bad examples of the LVDS connectors. In the right example, the differential pairs are the same length; in the left example, signals of the same differential pair have different lengths, thereby causing skew.

Figure 8. "Bad" (left) and "Good" (right) LVDS Connectors



Use the following guidelines while selecting connectors to be used for LVDS applications.

- Connectors must have low skew with matched impedance.
- Connectors with same length leads should be selected for lower skew and crosstalk.
- The two members of the same differential pair must be placed adjacent to each other on the connector.
- Ground pins should be placed between differential pairs.
- End pins of the connectors should preferably be grounded and must not be used for high-speed signals.
- All the unused pins of the connector should be properly terminated.

Summary

To benefit from the high speed and low noise of LVDS, designers must ensure that differential trace conductors, both on-board and going through connectors or cables, are closely coupled so as to have low noise and are well balanced for low skew and impedance matching.

Further Information

- Power Distribution Network Design Tool: www.altera.com/technology/signal/power-distribution-network/sgl-pdn.html

Document Revision History

Table 1 shows the revision history for this document.

Table 1. Document Revision History

Date	Version	Changes
September 2010	2.1	■ Minor text edit to Impedance Matching.
July 2010	2.0	■ Updated General PCB Guidelines. ■ Removed LVDS in APEX Devices, Figure 6. ■ Minor text edits.
July 2000	1.0	Initial release.

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训推荐课程列表： http://www.edatop.com/peixun/tuijian/

射频工程师养成培训课程套装

该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

课程网址：http://www.edatop.com/peixun/rfe/110.html

手机天线设计培训视频课程

该套课程全面讲授了当前手机天线相关设计技术，内容涵盖了早期的外置螺旋手机天线设计，最常用的几种手机内置天线类型——如 monopole 天线、PIFA 天线、Loop 天线和 FICA 天线的设计，以及当前高端智能手机中较常用的金属边框和全金属外壳手机天线的设计；通过该套课程的学习，可以帮助您快速、全面、系统地学习、了解和掌握各种类型的手机天线设计，以及天线及其匹配电路的设计和调试...

课程网址： http://www.edatop.com/peixun/antenna/133.html

WiFi 和蓝牙天线设计培训课程

该套课程是李明洋老师应邀给惠普 (HP)公司工程师讲授的 3 天员工内训课程录像，课程内容是李明洋老师十多年工作经验积累和总结，主要讲解了 WiFi 天线设计、HFSS 天线设计软件的使用，匹配电路设计调试、矢量网络分析仪的使用操作、WiFi 射频电路和 PCB Layout 知识，以及 EMC 问题的分析解决思路等内容。对于正在从事射频设计和天线设计领域工作的您，绝对值得拥有和学习！…

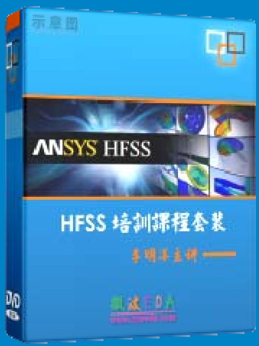
课程网址：http://www.edatop.com/peixun/antenna/134.html

CST 学习培训课程套装

该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

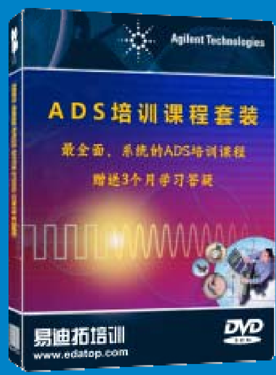
HFSS 学习培训课程套装



该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的 HFSS 培训教程套装，可以帮助您从零开始，全面深入学习 HFSS 的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

ADS 学习培训课程套装



该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com