

# **LVDS SerDes Gen I PCB and Interconnect Design-In Guidelines**

---



---

## **ABSTRACT**

The LVDS SerDes delivers multiple benefits to data communication and telecommunication applications. The LVDS serializer's function is to convert a wide parallel TTL bus into a smaller faster LVDS interface, and the deserializer's function is to recover the data and re-generate the wide TTL bus. LVDS SerDes include a smaller LVDS interface which saves PCB space, simplifying the task of PCB trace layout. The LVDS drivers are capable of driving long cables (up to 10 meters) at high speeds. The smaller cables and connectors are lower cost since they are physically smaller (up to 80% reduction in the number of required conductors/pins). LVDS supports higher bandwidth service with a reduction in EMI due to the differential scheme (odd mode), the reduced signal swing (only  $\pm 300$  mV), and the use of current mode drivers (soft transitions, and reduced spikes). To gain the maximum benefits of LVDS SerDes, high speed PCB and interconnect design is required. This application note focuses on the requirements of the PCB and interconnect to provide an error free, low emission LVDS interface.

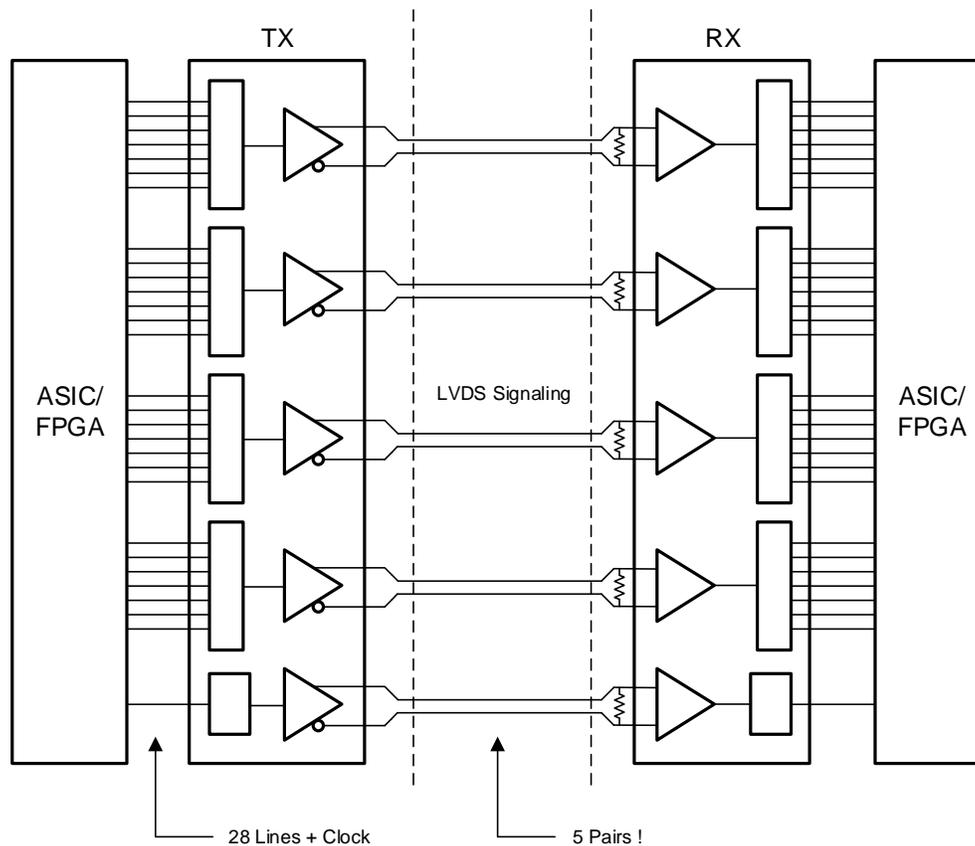
## **Contents**

1	Link Introduction .....	2
2	Interface Connections .....	4
3	Connectors .....	7
4	The Interconnect .....	7
5	Bit Mapping .....	8
6	MHz and Mbps .....	8
7	Summary .....	9
8	Generalized PCB Recommendations of LVDS and Link Applications .....	9

## **List of Figures**

1	Link System Block Diagram .....	2
2	TX and RX Location .....	3
3	LVDS PCB Layout .....	5
4	Differential Trace Spacing .....	6
5	Termination Layout Standard and Fly-by .....	7

## **List of Tables**

**Trademarks**
**1 Link Introduction**

**Figure 1. Link System Block Diagram**
**1.1 Function of the TX and RX**

The Transmitter (TX) accepts the wide parallel (21-bit or 28-bit) TTL bus and converts it to three or four higher speed serialized LVDS data streams. Applications may employ the full payload (21 or 28 bits) of the chipset or only a portion (i.e. 14, 16, 17, 18, 20 bit wide).

To operate the TX, only data and clock must be applied. The chip does not require control signals, as it provides transparent data transmission. Data on the TX inputs is strobed into the TX on the rising edge of the clock signal. The clock is then fed to a PLL type circuit that is used to generate strobes that internally clock out the serialized LVDS data streams. A powerdown pin is also supported. Asserting this pin shuts down the PLL and also puts the driver outputs into TRI-STATE. This mode disables the LVDS load loop current and also reduces ICC to  $\mu$ As, saving power when the link is not needed. The 3.3-V TX devices feature high impedance bus pins when the device is powered off.

The Receiver (RX) accepts the three or four high-speed serialized LVDS data streams and converts them back to a wide parallel TTL bus. The RX device is the complementary function to the TX device. Data on the RX output is strobed by the RXCLKout on the rising edge of the clock signal. The RXs utilize the rising edge of the LVDS Clock signal to strobe the data on the RX LVDS data inputs. A powerdown pin is also supported. Asserting this pin shuts down the PLL and also locks the RX outputs to the current state or low state. This mode saves power when the link is not needed. The 3.3-V RX devices also feature high impedance bus pins when the device is powered off. This feature enables multi-drop applications for single TX to multiple RX configurations.

## 1.2 Location of TX and RX on the PCB

The transmitter should be located as close to the connector as possible (see Figure 2). This is done to minimize the LVDS PCB overall trace length (see Section 2.3 on differential traces) and skew as well. Skew is generally proportional to length, thus a shorter interconnect has nominally less skew associated with it. If the TX is located within 2 inches of the connector, serpentine (trace length compensation to generate zero skew) is not required. If the TX is more than 2 inches away, equal length traces should be employed.

It is not a requirement of the TX device to be located adjacent to the ASIC/FPGA (bus controller) device. Recall the ASIC-TX interface is a slow speed bus (compared to the LVDS bus) and has fewer timing constraints. However, the connection between the ASIC and the TX should be designed such that the parallel bus traces are also of equal length to avoid excessive skew which could cause a set-up or hold violation at the TX input pins. In the extreme case (if electrical length is 1/2 rise time), signal termination may be required on the ASIC-TX interface. Note, the bus data lines are level sensitive only, while the clock line is edge sensitive. If the interconnect between the ASIC and the TX is long, (becomes a transmission line), termination for the clock signal should be considered. Input voltages applied to the TX input pins should remain within the VCC to GND range. Exceeding the ABS MAX ratings of the input pins increases noise in the device (and jitter) and can falsely trigger the ESD protection circuitry.

The receiver should also be located close to the connector as shown in Figure 2. This is done to minimize the LVDS PCB trace length. It is not a requirement of the RX device to be located adjacent to the destination controller (ASIC/FPGA). If the distance is greater than a few inches, or for high fanouts (more than 2), external buffering of the RX outputs should be employed. The connection between the controller and the RX should be designed such that the parallel data bus traces are of equal length to avoid skew which could cause a set-up or hold violation at the ASIC controller input pins.

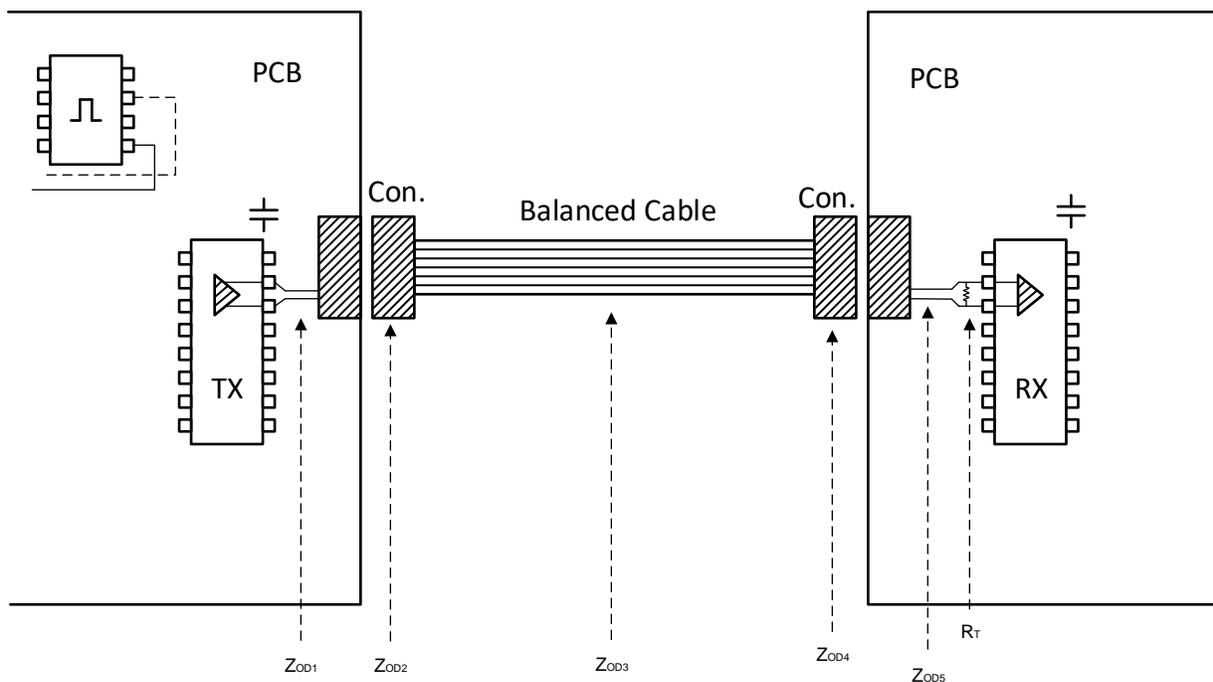


Figure 2. TX and RX Location

### 1.3 Device Ground and Power

The TX and RX are high speed — high performance devices. The devices include a PLL type circuit that operates at  $f$  or  $3.5x f$  MHz (device specific, where  $f$  is the clock frequency) and LVDS drivers and receivers that toggle in the hundreds of Megabits per second range (up to 462 Mbps at a 66MHz clock rate!). This requires a solid ground and power distribution reference for the device. VCC noise should not exceed 20 mVpp. If it does, separated power and ground planes should be used for the PLLVCC and GND pins. To enable good power/ground reference a minimum of a 4 layer PCB is recommended. Two sided boards or flex circuit are generally not recommended for placement of the TX or RX devices. EMI emission control starts with solid power and ground planes.

### 1.4 Device Bypass

The TX and RX have three separate VCC (power supply) and GND references. These are for the digital logic, LVDS, and PLL portions of the chip. To obtain supply noise of less than 100 mVpp; close bypassing is required. Do not locate bypass capacitors at the end of small traces, rather connect them adjacent to the device pins as close as possible. Bulk capacitance of 4.7  $\mu$ F to 10  $\mu$ F should also be near by. Wide traces with multiple power/GND via (2 are recommended) should be employed on bypass connections. The ideal case would use 0.1  $\mu$ F//0.01  $\mu$ F//0.001  $\mu$ F capacitors on each supply pin.

If space is restricted do not eliminate the PLL bypassing as this is most critical of low noise operation. 0805 or 1206 chip capacitors are recommended as they offer the lowest inductance and can be mounted very close to the device pins. With this type of layout, parasitic inductance will be in the 1.5 nH to 2 nH range, whereas locating the capacitor at the end of a short narrow trace can have an inductance as high as 15 nH! Using multiple capacitors in parallel provides good bypassing across a wider range of frequency, also the ESR is lowered and a low impedance at high frequency is provided.

## 2 Interface Connections

### 2.1 ASIC — TX Interface

The connection between the ASIC controller and the TX device should utilize standard PCB design techniques. The data bus and clock lines should all be of equal length in order to prevent any skew being introduced between clock and data. As with any clock line, signal quality is of concern. Avoid unnecessary via, sharp bends or other discontinuities. A direct point-to-point link is best for the clock and its signal quality. Other clock signals should be kept away from the clock and data lines to avoid unwanted coupling. Provision for termination may be required on the clock signal at the TX input.

If the ASIC has drive level options (6 mA–8 mA or 12 mA modes, device specific), testing has shown that operating the clock in high drive and the data in low offers the best balance between clock signal quality (sharp) and noise generation from the ASIC data output pins. Unused TX input pins should be tied to GND on the 5 V series and may be left floating or tied to GND on the 3.3 V series to prevent unwanted switching and the lowest chip power dissipation. The 3.3 V devices feature internal pull down resistors on the inputs.

### 2.2 RX — ASIC Controller Interface

The connection between the RX and the ASIC controller should utilize standard PCB design techniques. The data lines and clock should all be of equal length in order to prevent any skew being introduced between clock and data. As with any clock line, signal quality is of concern. Avoid unnecessary vias, sharp bends or other discontinuities. A direct point to point link is best for the clock and its signal quality. If a high fanout of the RX output is required (more than 2 loads), external buffering is recommended.

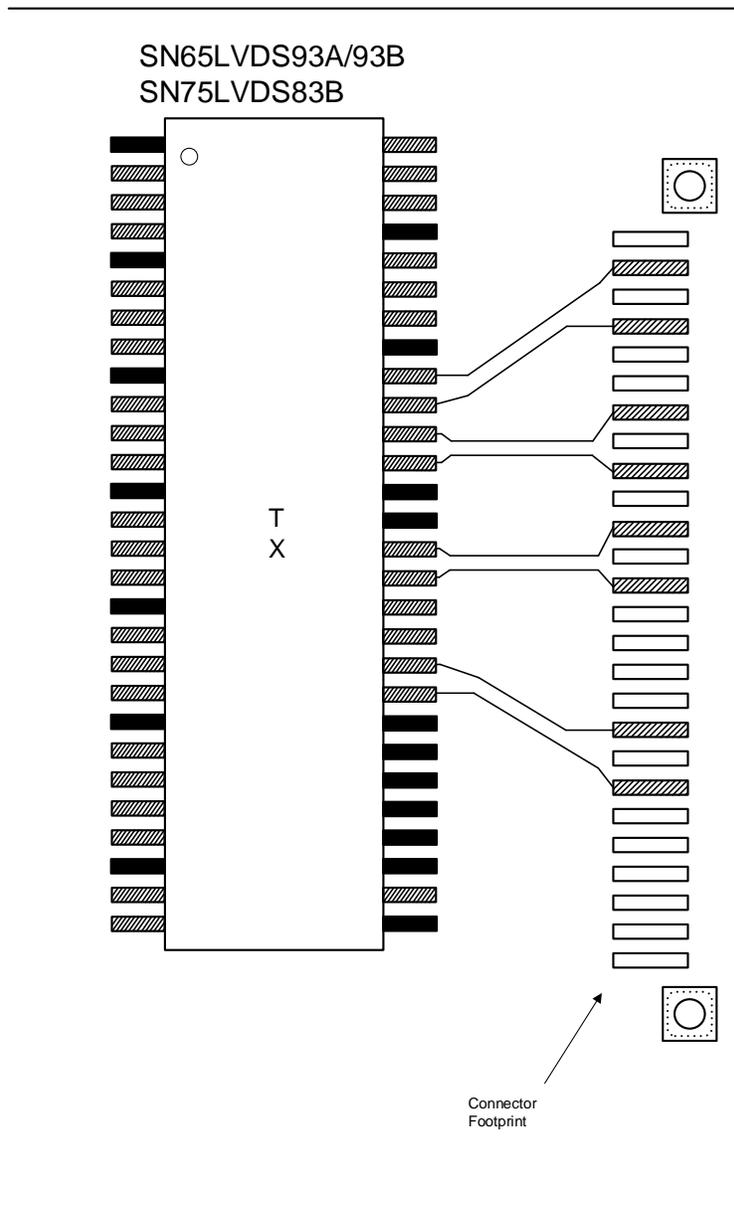


Figure 3. LVDS PCB Layout

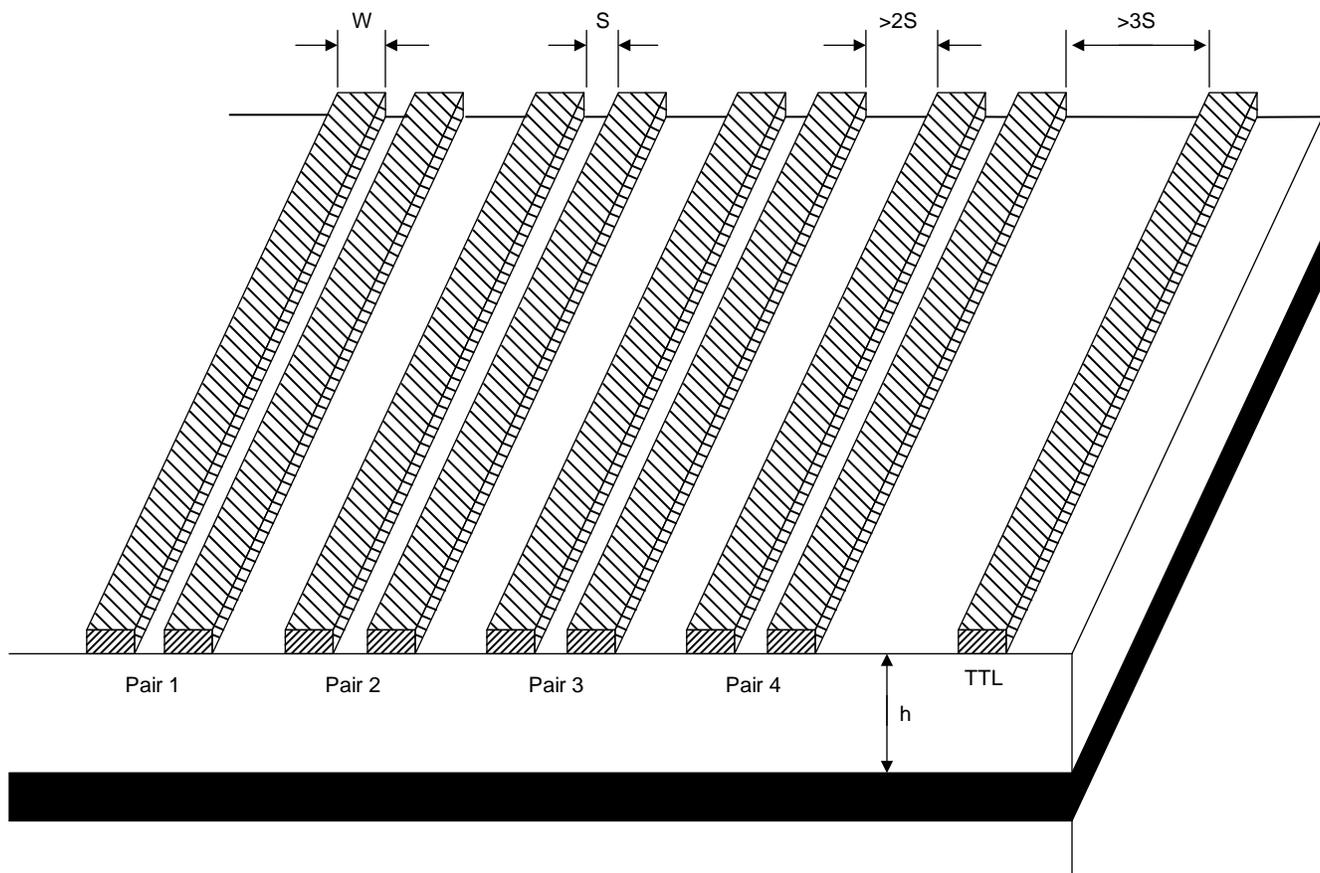
### 2.3 LVDS Device — Connector Interface

The traces that connect the TX LVDS outputs to the connector and the connector to the RX inputs should be minimized in length as discussed above in the section on TX/RX location. In addition TTL/CMOS single-ended lines should be located on a different signal layer or kept away (at least “3S”) from the LVDS lines to minimize any coupling of noise onto the LVDS lines and out onto the interconnect as shown in Figure 3. Guard ground traces may be placed between the LVDS lines and the other signals to further isolate the two, they should be at least 2S away from the closely-coupled differential pair. Corruption of LVDS data is not an issue due to the receiver’s common mode rejection; minimizing common mode noise (EMI) is the reason for the spacing and isolation. Unused LVDS driver output pins should be left open, as this will minimize power dissipation. Unused RX inputs should also be left open (not terminated). The use of true differential traces further enhances the noise rejection capabilities of differential data transmission and reduces / limits the generation of unwanted emissions.

Differential traces are recommended for use on the LVDS outputs of the TX and inputs to the RX. They differ only slightly from standard PCB traces. The distance between the two signal traces (“S”) that compose the differential pair is also controlled (minimized). This distance is critical to specify correctly as it is related to both the differential-mode characteristic impedance of the trace pair and also related to the differential noise margin of the system.

Standard traces are laid out for  $50\Omega$  impedance. Two traces separated far apart from each other will present a differential mode characteristic impedance of twice the single-ended impedance, but will not provide the maximum noise rejection benefit and defeat the differential signaling. The spacing between the traces should be kept to a minimum to maximize the differential noise rejection (differential noise margin is equal to the minimum signal swing less the maximum receiver thresholds ( $250\text{ mV} - 100\text{ mV} = 150\text{ mV}$ )). This minimum distance between the traces of a pair assures that any external noise coupled onto the pair will be seen as common mode, and rejected by the receivers. To adjust trace impedance set “S” to a minimum spacing between metal lines, and adjust the width of the trace to the desired impedance (see AN-905 for equations). When the two lines are close, the differential impedance is no longer twice the single-ended impedance (it is a factor of  $<2$ ). The distance between adjacent LVDS differential pairs should be at least  $2S$  or greater as shown in Figure 3. The electrical length of the LVDS traces should be matched to minimize any skew created in the path between TX and RX. Lastly, it is desirable to have  $S < h$ , again ensuring that maximum coupling occurs between the pair and not with the plane below.

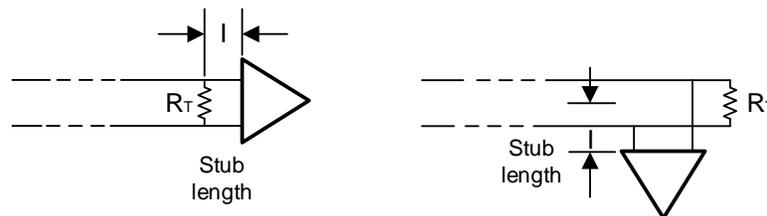
Microstrip traces (outside layer) are usually employed for this interconnect. The microstrip geometry and the fact that the traces are on the outer layer enable higher impedances more easily. It is also possible to design the interconnect without any via, thus a better signaling environment. On the down side, stripline (embedded) traces offer greater shielding due to their encasement. Either microstrip or stripline traces can be used, as long as they are matched to the cable’s differential-mode characteristic impedance. This is shown in Figure 2 where  $Z_{OD1}$  represents the TX PCB impedance and  $Z_{OD3}$  is the cable (interconnect) impedance. It is recommended that all impedances should be within 10% of the target impedance (typically  $100\Omega$ ) to minimize any reflections. Note that a 10% difference in impedance will create a reflection of 5%. Reducing the magnitude of reflections will lower the EMI of the system.



**Figure 4. Differential Trace Spacing**

## 2.4 Termination at the RX Inputs

The use of a termination resistor is required. Due to the high speed edge rates of LVDS drivers a matched termination will prevent the generation of any signal reflections, and reduce EMI. Termination is typically implemented with a single surface mount resistor connected across the signal pair as close to the receiver inputs as possible (no more than 1/2 inch away if the termination is placed in front of the receiver inputs). If the resistor is too far away from the receiver inputs, the line between the resistor and the receiver input takes on transmission line behavior and reflections may occur at the receiver inputs defeating the purpose of the termination resistor altogether! Also the use of an external resistor allows for a variety of media to be used. Since characteristic impedance is media dependent, the proper termination resistance may be employed to match the particular application. The resistor should be selected to within 2% of the nominal differential impedance (100Ω) of the media. If PCB space is tight, a fly-by termination may also be used. The two termination methods are shown in Figure 5. In either case the resulting stub should be kept as short as possible.



**Figure 5. Termination Layout Standard and Fly-by**

## 3 Connectors

For intra-box applications (board to board in a box) the connector is typically very small since it is internal to the box and is only connected upon assembly of the system. These connectors have the advantage that they do not adversely effect the signal quality greatly, since they are electrically small and present more of a small lumped load to the signal. Larger connectors used between boxes (box to box application) are more critical since they are electrically long and may react as a transmission line segment. High performance connectors are available that provide controlled impedance and matched electrical length of the pins (no skew). The 3M MDR system is one such cabling system that meets the needs of LVDS applications for cable and connector requirements. It offers a zero skew SMT connector, controlled impedance, low crosstalk, and very low skew (see reference in the appendix). Other connectors that have been employed include standard SCSI connectors, and also DB15 connectors to name a few.

### 3.1 Connector Pinout

Connector Pinout is application dependent. On multiple row connectors, the pair should be routed on adjacent pins in the same row to minimize skew within the pair. Another option is to employ single row low skew surface mount connectors. A signal common (ground) path should also be provided near the pairs to provide a low impedance near by return path for common mode noise picked up along the cable. This will help to reduce unwanted EMI.

## 4 The Interconnect

LVDS drivers and receivers are intended to operate on a wide variety of media. Depending upon the system's needs the media may vary due to a number of parameters including: length of interconnect, amount of shielding required, physical dimensions of the system, and of course cost. LVDS SerDes (LVDS) has been demonstrated on flat ribbon cable, FEC (flex) interconnect, shielded twisted pair, and twin axial cables. An important parameter to understand is the differential-mode characteristic impedance of the media. For many common cables this is typically about 100Ω.

As discussed above to minimize any common mode noise generation the TX PCB interconnect, the interconnecting media, the RX PCB interconnect, and the termination should all match (within 10%) in characteristic impedance (differential mode). Electrical length of the data and clock signals should be of equal length. Conductors of different length composing a pair will cause a modulation of common mode and radiate more. Pairs of different length will impact the receivers data recovery by impacting the correct

strobing of data. Remember that the timing of the LVDS data line is 7X tighter than of the TTL bus. The amount of tolerable skew between any two conductors is clock speed dependent, but should be kept to less than 100 ps–400 ps (data rate dependent, 66 MHz–20 MHz) with the Link chips. On a typical high performance cable (3M MDR) skew is specified at only 30 ps/meter. For flex circuit interconnects, similar design recommendations as discussed in [Section 2.3](#) above should be employed. Even though LVDS operates as true odd-mode differential drivers, a signal common (Ground) connection is required between the two systems to establish a common mode return path. The bulk of the LVDS load current is returned to the driver within the pair as LVDS is a true odd mode differential driver. This is very important for emission reasons, as the closely-coupled differential pair creates a small ring antenna. However, for common mode return, a signal common connection of low impedance is required for low emission operation. Typically assigning one pair (two conductors) is sufficient for this purpose (as it has 1/2 the DCR (DC resistance) of a single line). Shield ground references should be tied off to quiet ground references, typically frame ground at each end if employed. LVDS SerDes (LVDS) may be used on a wide variety of media (flat cable, twin-ax, standard twisted pair,...cables). Length is a factor of cable performance, balance, skew, and clock speed, thus it is application dependent.

General guidelines for different types of cable are given next: for very short distance applications (less than 0.3 meter) flat cable or flex circuit has been employed. For medium distance applications (less than 7 meters) standard twisted pair cables have been employed. For application exceeding 5 meters, typically twin-ax cables have been employed due to their inherent low skew between pairs. Greater than 10 meter applications are possible, however sampling margin, skew, and impedance control must be carefully reviewed.

## 5 Bit Mapping

The bit mapping for the 21 and 28 bit chipsets are shown in the respective datasheets. Bit mapping is the same for all 21 bit parts (40, 66, and 85MHz families). Bit mapping is the same for all 28 bit parts (40, 66, 85MHz families). Note that the 21 and 28 bit part families are mapped differently. Mixing 3.3V and 5V devices is also possible, since all devices adhere to the LVDS interface standard. If less than 21 or 28 bits are required, multiple options to disable particular bits are possible. The following options are possible: If there are 7 unused bits, then it is possible to utilize only 2 LVDS data streams (21 bit chipset example). This will lower the size of the interconnect required. Unused TX inputs should be left open on the 3.3V family (and tied to GND on the 5V family of TXs). The unused LVDS channel outputs should also be left open to minimize current (power dissipation). Unused RX inputs may also be left open, as these inputs feature a failsafe feature that pulls the plus input high and the minus input low, preventing unwanted transitions. Unused RX outputs should also be left open to minimize power dissipation. If there are only one to seven unused bits in the application, then bits on each channel may be disabled. This can be used to enhance signal quality on the cable as well for long cable applications. By disabling two data bits in the data stream (tie one HIGH and the other LOW), inserts guaranteed transitions on the data channel and improves (opens) the eye pattern (reduces ISI distortion of NRZ data).

## 6 MHz and Mbps

Mega Hertz and “Mega bits per second” are two terms that are commonly confused. Mega Hertz (MHz) is a measure of the clock period applied to the TX clock input. Mega bit per second is the information rate, and for NRZ data is the measure of a single bit width. It is also important to indicate where the bit rate is being measured. For the Link chipset, there are three different possible bps values. These are:

1. The bit rate on a single TX input
2. The bit rate on a single LVDS data channel
3. The bit rate on the entire link (3 or 4 LVDS channels combined)

For example, with a 66MHz clock, and with data strobed on the rising edge, the bit rate on a single TX input is 66Mbps. The bit rate on a LVDS channel is 7X the input rate, or 462Mbps. The bit rate on the entire 28 bit LVDS SerDes is 4X the LVDS channel rate, or 1.848Gbps.

## 7 Summary

High speed PCB and Interconnect design practices should be employed to ensure an error free, low emission design for the LVDS SerDes devices. The interconnect is a transmission line due to the high speed edge rates of the LVDS signals (500 ps typical). Tight skew control is required to minimize emissions and proper data recovery at the RX devices. Matching impedances within 10% is recommended to reduce the creation of reflections (even mode) along the interconnect/cable.

## 8 Generalized PCB Recommendations of LVDS and Link Applications

LVDS features fast edge rates, therefore the interconnect between transmitters and receivers will act as a transmission line. The PCB traces that form this interconnect must be designed with care. The following general guidelines should be adhered to:

- Hand route or review very closely auto-routed traces.
- Locate the Transmitters and Receivers close to the connectors to minimize PCB trace length for off PCB applications.
- LVDS traces should be designed for differential impedance control (space between traces needs to be controlled).
- Minimize the distance between traces of a pair to maximize common mode rejection.
- Place adjacent LVDS trace pairs at least twice (2S) as far away (as the distance between the conductors of the pair).
- Place TTL/CMOS (large dV signals) far away from LVDS, at least three times (3S) away or on a different signal layer.
- Match electrical length of all LVDS lines.
- Keep stubs as short as possible.
- Avoid crossing slots in the ground plane.
- Avoid 90° bends (use two 45s).
- Minimize the number of via on LVDS traces.
- Maintain equal loading on both traces of the pair to preserve balance.
- Match impedance of PCB trace to connector to media (cable) to termination in order to minimize reflections (emissions) for cabled applications (typically 100Ω differential mode impedance).
- Select a termination resistor to match the differential mode characteristic impedance of the interconnect, 2% tolerance is recommended.
- Locate the termination within 1/2 (<1) inch of the receiver inputs if not using a fly-by termination method.
- Use surface mount components to minimize parasitic L and C for bypass caps and termination resistors.
- Use a 4 layer PCB (minimum). Bypass each LVDS package at the device pin (Bulk bypass nearby also) with parallel capacitors (0.1 μF//0.01 μF//0.001 μF) on each of the supply pins (VCC, LVDS VCC, and PLL V CC).

## IMPORTANT NOTICE FOR TI DESIGN INFORMATION AND RESOURCES

Texas Instruments Incorporated ("TI") technical, application or other design advice, services or information, including, but not limited to, reference designs and materials relating to evaluation modules, (collectively, "TI Resources") are intended to assist designers who are developing applications that incorporate TI products; by downloading, accessing or using any particular TI Resource in any way, you (individually or, if you are acting on behalf of a company, your company) agree to use it solely for this purpose and subject to the terms of this Notice.

TI's provision of TI Resources does not expand or otherwise alter TI's applicable published warranties or warranty disclaimers for TI products, and no additional obligations or liabilities arise from TI providing such TI Resources. TI reserves the right to make corrections, enhancements, improvements and other changes to its TI Resources.

You understand and agree that you remain responsible for using your independent analysis, evaluation and judgment in designing your applications and that you have full and exclusive responsibility to assure the safety of your applications and compliance of your applications (and of all TI products used in or for your applications) with all applicable regulations, laws and other applicable requirements. You represent that, with respect to your applications, you have all the necessary expertise to create and implement safeguards that (1) anticipate dangerous consequences of failures, (2) monitor failures and their consequences, and (3) lessen the likelihood of failures that might cause harm and take appropriate actions. You agree that prior to using or distributing any applications that include TI products, you will thoroughly test such applications and the functionality of such TI products as used in such applications. TI has not conducted any testing other than that specifically described in the published documentation for a particular TI Resource.

You are authorized to use, copy and modify any individual TI Resource only in connection with the development of applications that include the TI product(s) identified in such TI Resource. NO OTHER LICENSE, EXPRESS OR IMPLIED, BY ESTOPPEL OR OTHERWISE TO ANY OTHER TI INTELLECTUAL PROPERTY RIGHT, AND NO LICENSE TO ANY TECHNOLOGY OR INTELLECTUAL PROPERTY RIGHT OF TI OR ANY THIRD PARTY IS GRANTED HEREIN, including but not limited to any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI products or services are used. Information regarding or referencing third-party products or services does not constitute a license to use such products or services, or a warranty or endorsement thereof. Use of TI Resources 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.

TI RESOURCES ARE PROVIDED "AS IS" AND WITH ALL FAULTS. TI DISCLAIMS ALL OTHER WARRANTIES OR REPRESENTATIONS, EXPRESS OR IMPLIED, REGARDING TI RESOURCES OR USE THEREOF, INCLUDING BUT NOT LIMITED TO ACCURACY OR COMPLETENESS, TITLE, ANY EPIDEMIC FAILURE WARRANTY AND ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND NON-INFRINGEMENT OF ANY THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

TI SHALL NOT BE LIABLE FOR AND SHALL NOT DEFEND OR INDEMNIFY YOU AGAINST ANY CLAIM, INCLUDING BUT NOT LIMITED TO ANY INFRINGEMENT CLAIM THAT RELATES TO OR IS BASED ON ANY COMBINATION OF PRODUCTS EVEN IF DESCRIBED IN TI RESOURCES OR OTHERWISE. IN NO EVENT SHALL TI BE LIABLE FOR ANY ACTUAL, DIRECT, SPECIAL, COLLATERAL, INDIRECT, PUNITIVE, INCIDENTAL, CONSEQUENTIAL OR EXEMPLARY DAMAGES IN CONNECTION WITH OR ARISING OUT OF TI RESOURCES OR USE THEREOF, AND REGARDLESS OF WHETHER TI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

You agree to fully indemnify TI and its representatives against any damages, costs, losses, and/or liabilities arising out of your non-compliance with the terms and provisions of this Notice.

This Notice applies to TI Resources. Additional terms apply to the use and purchase of certain types of materials, TI products and services. These include; without limitation, TI's standard terms for semiconductor products (<http://www.ti.com/sc/docs/stdterms.htm>), [evaluation modules](#), and [samples](http://www.ti.com/sc/docs/sampterm.htm) (<http://www.ti.com/sc/docs/sampterm.htm>).

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265  
Copyright © 2018, Texas Instruments Incorporated