

FPD-Link PCB and Interconnect Design-In Guidelines

National Semiconductor
 Application Note 1085
 John Goldie
 June 1999



FPD-Link PCB and Interconnect Design-In Guidelines

INTRODUCTION

The Flat Panel Display Link (FPD-Link) Transmitter's function is to convert a wide parallel TTL bus into a smaller faster LVDS interface, and the Receiver's function is to recover the data and re-generate the wide TTL bus. The system block diagram is shown in Figure 1. FPD-Link delivers multiple benefits to Notebook and Monitor LCD applications. These include the following: The smaller LVDS interface is simpler to route between the motherboard and the LCD panels (i.e., through the hinge). The LVDS drivers are capable of driving long cables at high speeds for monitor applications. The

smaller cables and connectors (up to 80% reduction in the number of required conductors) are lower cost since they are physically smaller. LVDS supports higher bandwidth service with a reduction in EMI due to the differential scheme (odd mode), the reduced signal swing (only ~300 mV), and the use of current mode drivers (soft transitions, and reduced spikes). To gain the maximum benefits of LVDS and FPD-Link 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.

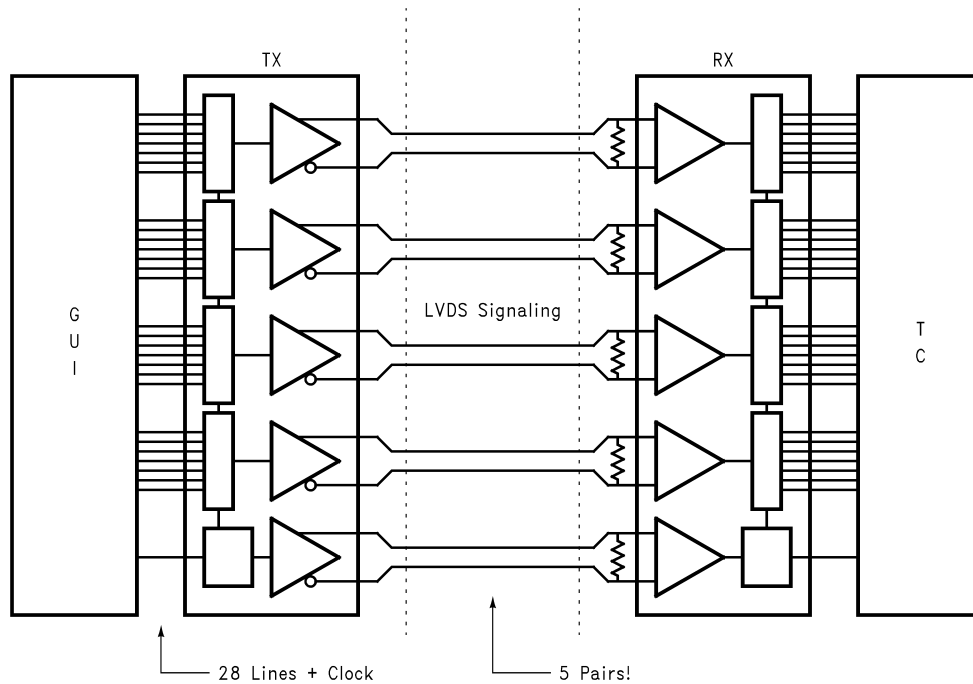


FIGURE 1. Link System Block Diagram

FUNCTION OF THE TX

The Transmitter (TX) accepts the wide parallel (21-bit or 28-bit) TTL bus and converts it to 3 or 4 higher speed serialized LVDS data streams. To operate the TX only data and clock must be applied. The chip does not require control signals, as it provides transparent data transmission. Depending upon the device used, the data on the TX inputs is strobed into the TX on either the rising or falling edge of the clock signal. The particular edge is noted within the device ID with "R" or "F" respectively. Some devices provide a pin to select R or F operation (e.g., DS90C383/A). 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 power-

down 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 I_{CC} to μ As, saving power when the display is not needed.

LOCATION OF TX ON PCB

The transmitter should be located as close to the interface as possible (see Figure 2). This is done to minimize the PCB LVDS overall trace length (see following section on differential traces) and thus skew as well. Skew is generally proportional to length, thus a shorter interconnect nominally has less skew associated with it. If the TX is located within 2 inches of the connector, mitering (trace length compensation

TRI-STATE® is a registered trademark of National Semiconductor Corporation.

AN-1085

to generate zero skew) is not required. If the TX is more than 2 inches away, mitering should be employed. Also, it is not a requirement of the TX device to be located adjacent to the GUI device. Recall the GUI-TX interface is a slow speed bus (compared to the LVDS bus) and has less timing constraints. However, the connection between the GUI and the TX should be designed such that the parallel RGB traces are also of equal length to avoid excessive skew and a resulting

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 GUI interface. Note, the RGB data lines are level sensitive only, while the clock line is edge sensitive. If the interconnect between the GUI and the TX is long, (becomes as a transmission line), termination for the clock (FPSHIFT) signal should be considered.

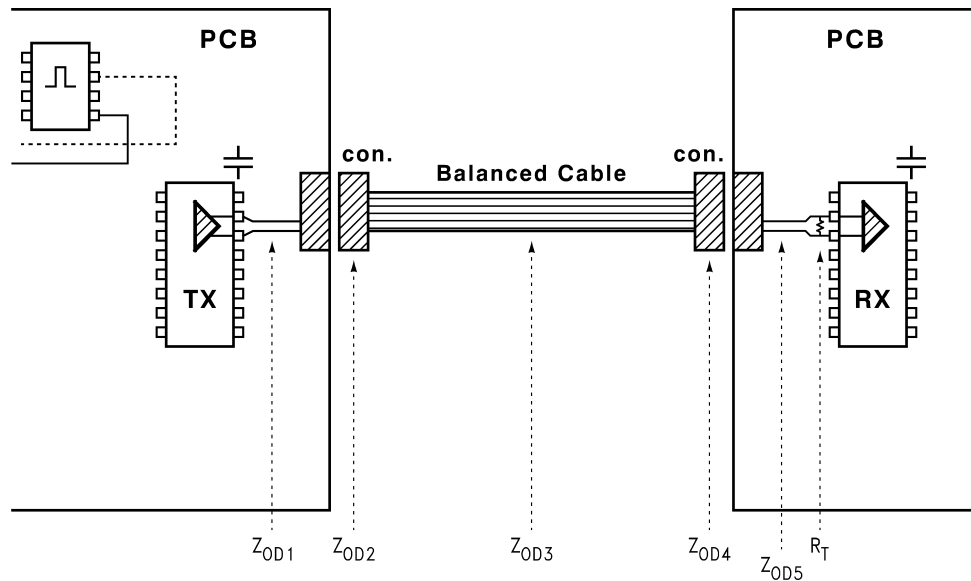


FIGURE 2. TX and RX Location

TX GROUND AND POWER

The TX is a high speed — high performance device. The device includes a PLL type circuit that operates at f or $3.5Xf$ MHz (device specific, where f is the clock frequency) and LVDS drivers that toggle in the hundreds of Megabits per second range (up to 455 Mbps 65MHz clock rate!). This requires a solid ground and power distribution reference for the device. V_{CC} noise should not exceed 100 mV_{pp} . If it does, separated power and ground planes should be used for the PLL V_{CC} and PLLGND 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. If the TX is an option to the application and is intended to be located on a small daughter card, the daughter card should also be 4 layers and employ multiple connections to power and ground (low impedance connection).

TX BYPASS

The TX includes three separate V_{CC} (power supply) and GND references. These are for the digital logic, LVDS drivers, and PLL portions of the chip. To obtain supply noise of less than 100 mV_{pp} , close by-passing is required. Do not locate by-pass capacitors at the end of small traces, rather connect them adjacent to the device pins as close as possible. Bulk capacitance of $4.7\text{ }\mu\text{F}$ to $10\text{ }\mu\text{F}$ should also be near by. Wide traces with multiple power/GND via (2 are rec-

ommended) should be employed on bypass connections. The ideal case would use $0.1\text{ }\mu\text{F}/0.01\text{ }\mu\text{F}/0.001\text{ }\mu\text{F}$ capacitors on each supply pin. If space is restricted do not eliminate the PLL by-passing 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, where as 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.

GUI — TX INTERFACE

The connection between the GUI controller and the TX device should utilize standard PCB design techniques. The data (R, G, and B), control, 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 GUI 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 GUI 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 (see datasheets).

TX—CONNECTOR INTERFACE

The traces that connect the TX LVDS outputs to the connector should be minimized in length as discussed above in the section on TX 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, once again 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.

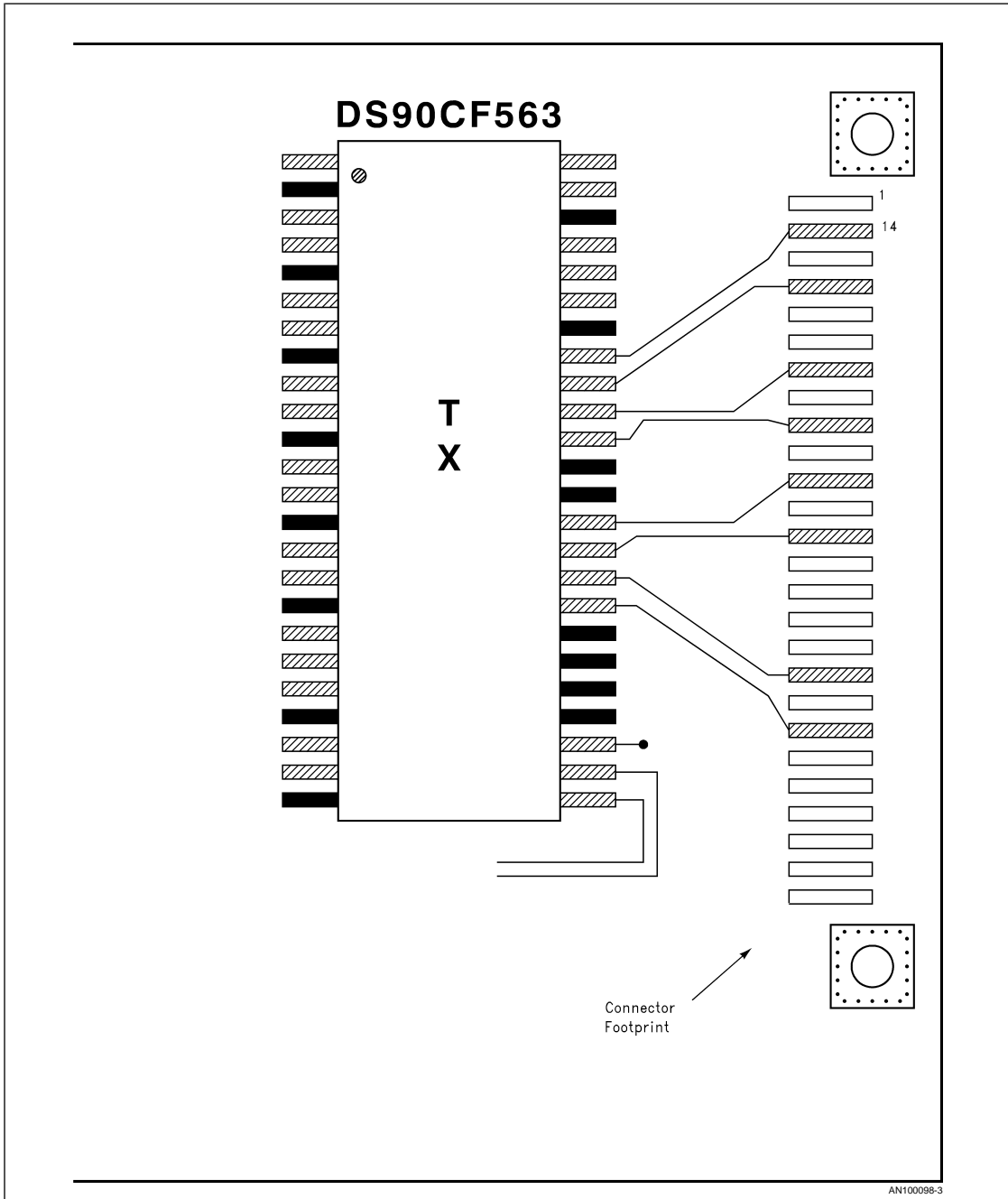


FIGURE 3. LVDS PCB Layout

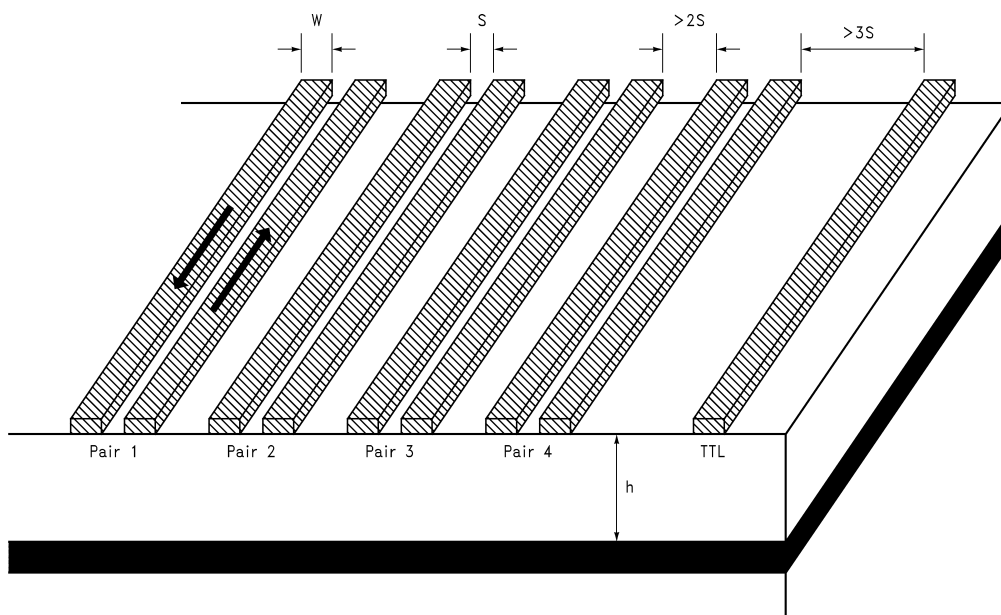
AN100098-3

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 output of the TX (and input to the receiver also). 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 50Ω 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 mV–100 mV = 150 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 im-

pedance 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 4. The distance between an LVDS signal and a large swing TTL or CMOS signal should be $3S$ or more, or even better the TTL/CMOS signaling should be located on a different signal plane. 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.



AN100098-4

FIGURE 4. Differential Trace Spacing

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 100Ω) 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.

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, dimensions of the hinge that the cable must pass through

(notebook application), and of course cost. FPD-Link (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 (see cable datasheet). 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–350 ps (data rate dependent, 65 MHz–32.5 MHz). 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 the “TX—connector interface” section 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 unlike some less superior solutions. This is very important for emission reasons, as the closely-coupled differential pair creates a small ring antenna, compared to a ON/OFF differential driver that returns current via the PWR or GND connections (thus creating a larger 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 of a single line). Shield ground references should be tied off to quiet ground references, typically frame ground at each end if employed. FPD-Link (LVDS) may be used on a wide variety of media from flat cable to twin-ax to standard twisted pair cables. Length is a factor of cable performance, balance, and skew, thus application dependent.

CONNECTORS

For intra-box applications (motherboard to panel in a notebook) 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 affect 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 (desktop PC to external monitor application) are more critical since they are electrically long and may react as a transmission line segment.

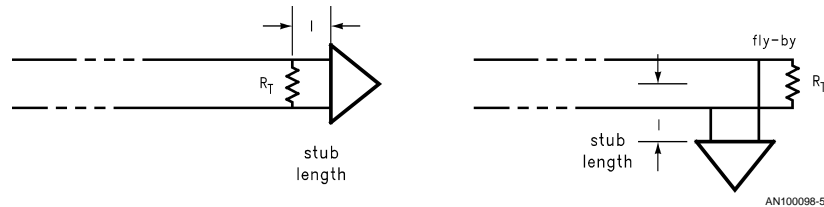


FIGURE 5. Termination Layout— Standard and Fly-By

Note: The complete interconnect between the TX output and the RX input is a high speed transmission line. Driver edge rates are in the 300 ps–700 ps range. The TX PCB traces, connectors, cable media, and RX PCB traces are all part of the system. Their impedances should be matched within 10% to maintain signal quality and generate the lowest amount of emissions. There are 4–6 impedances to consider, depending upon the application (the size of the connector). These are shown in Figure 2 as Z_{OD1} , Z_{OD2} , Z_{OD3} , Z_{OD4} , Z_{OD5} , and R_T . Also, minimize extra transitions (board or layer transitions, etc.) here to make this total interconnect as clean as possible.

FUNCTION OF THE RX

The Receiver (RX) accepts the 3 or 4 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. Depending upon the device used, the data on the RX output is strobed by the RxclkOUT on either the rising or falling edge of the clock signal. The particular edge is noted within the device ID with an “R” or “F” respectfully. All devices (R & F) utilize the rising edge of the LVDS Clock signal to strobe the data on the RX LVDS data inputs. A power-down pin is also supported. Asserting this pin shuts down the PLL and also locks the RX outputs to the current state or low state (device specific, see datasheets). This mode saves power when the display is not needed.

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 the application 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.

TERMINATION

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.

LOCATION OF RX ON PCB/PANEL SIDE

The receiver should be located close to the interface connector as shown in Figure 2. This is done to minimize the PCB LVDS trace length. (See the section on differential traces.) It is not a requirement of the RX device to be located adjacent to the timing controller. The connection between the timing controller and the RX should be designed such that the parallel RGB traces are of equal length to avoid skew and a resulting set-up or hold violation at the timing controller input pins.

CONNECTOR—RX INTERFACE

The PCB traces that connect the LVDS lines from the connector to the termination resistor and RX inputs should be minimized in length as discussed above in the section on TX Location for the same reasons. In addition any TTL/CMOS single-ended line should be located on a different signal layer or kept away from the LVDS lines to limit any coupling of noise onto the LVDS lines and back out onto the interconnect. Guard ground traces may be placed between the LVDS lines and other signals to further isolate the two. 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 isolation. The use of true differential traces further enhances the noise rejection capabilities of differential data transmission. Differential traces are recommended for use on the LVDS input of the RX (and output of the drivers). They differ only slightly from standard PCB traces. The distance between the two signal traces that compose the differential pair is also controlled. This distance is critical to specify correctly as it is related to both the differential-mode characteristic impedance of the pair and also related to the differential noise margin of the system and emissions. Standard traces are laid out for 50Ω 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 mV–100 mV = 150 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, and adjust the width of the trace to the desired impedance (see AN-905 for equations). The distance between adjacent LVDS differential pairs should be at least 2S or greater. The distance between a LVDS signal and a large swing TTL or CMOS signal should be 3S or more, or even better the TTL/CMOS signaling should be located on a different signal plane. The electrical length of the LVDS traces should be matched to minimize any skew created in the path between TX and RX. (See figures in the TX section.)

RX GROUND AND POWER

The RX is a high speed high performance device. The device includes a PLL circuit that operates at f (where f is the clock frequency) MHz and accepts LVDS data in the hundreds of Megabits per second range. This requires a solid ground and power distribution reference to the device. V_{CC} noise should not exceed 100 mV_{pp}. To enable good power/ground reference a minimum of a 4 layer PCB is required.

Two sided boards or flex circuit are generally not recommended. EMI emission control starts with solid power and ground planes.

RX BYPASS

The RX includes three separate V_{CC} (power supply) and GND references. These are for the digital logic, LVDS receivers, and PLL portions of the chip. To obtain supply noise of less than 100 mV_{pp} close by-passing is required. Do not locate by-pass capacitors at the end of small traces, rather connect them adjacent to the device pins as close as possible. Bulk capacitance of 4.7 μF to 10 μF 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 μF//0.01 μF//0.001 μF capacitors on each supply pin. If space is restricted do not eliminate the PLL by-passing 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, where as 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.

RX—TIMING CONTROLLER INTERFACE

The connection between the RX and the timing controller should utilize standard PCB design techniques. The data lines (R, G, and B), control, 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 via, sharp bends or other discontinuities. A direct point to point link is best for the clock and its signal quality.

RECOMMENDED COLOR BIT MAPPING

A recommended bit mapping is provided in *Tables 1, 2* and also graphically shown in *Figure 6*. *Table 1* lists the RGB bit numbers, TX input pin name, TX input pin number, RX output pin name, and RX output pin number for 8-bit color mapping. This is recommended for use with the DS90C383/A and DS90CF384 (56-lead TSSOP package) devices. *Table 2* lists the same information for 6-bit color mapping. This is recommended for use with the DS90C363/A and DS90CF364 (48-lead TSSOP package) devices. Note that this mapping allows for direct inter-operation of 8-bit color TX with a 6-bit RX device. The additional resolution bits for 8-bit graphics are located on LVDS data channel 3 (TxOUT3+ and TxOUT3–) and are the **LSBs**.

TABLE 1. Recommended 8-Bit Color Mapping (for 56L Devices)

Color Bit	TX Input Name	TX Pin Number	RX Input Name	RX Pin Number
R0-LSB+2	TxIN0	51	RxOUT0	27
R1	TxIN1	52	RxOUT1	29
R2	TxIN2	54	RxOUT2	30
R3	TxIN3	55	RxOUT3	32
R4	TxIN4	56	RxOUT4	33
R5-MSB	TxIN6	3	RxOUT6	35
R6-LSB	TxIN27	50	RxOUT27	7

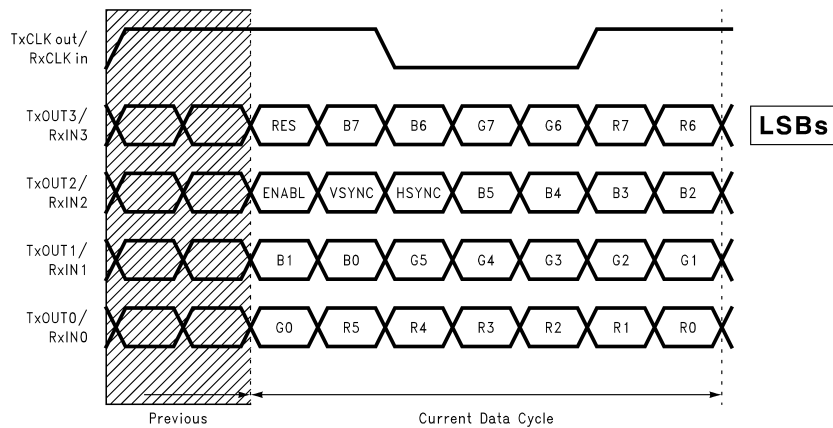
TABLE 1. Recommended 8-Bit Color Mapping (for 56L Devices) (Continued)

Color Bit	TX Input Name	TX Pin Number	RX Input Name	RX Pin Number
R7-LSB+1	TxIN5	2	RxOUT5	34
G0-LSB+2	TxIN7	4	RxOUT7	37
G1	TxIN8	6	RxOUT8	38
G2	TxIN9	7	RxOUT9	39
G3	TxIN12	11	RxOUT12	43
G4	TxIN13	12	RxOUT13	45
G5-MSB	TxIN14	14	RxOUT14	46
G6-LSB	TxIN10	8	RxOUT10	41
G7-LSB+1	TxIN11	10	RxOUT11	42
B0-LSB+2	TxIN15	15	RxOUT15	47
B1	TxIN18	19	RxOUT18	51
B2	TxIN19	20	RxOUT19	53
B3	TxIN20	22	RxOUT20	54
B4	TxIN21	23	RxOUT21	55
B5-MSB	TxIN22	24	RxOUT22	1
B6-LSB	TxIN16	16	RxOUT16	49
B7-LSB+1	TxIN17	18	RxOUT17	50
ENABL	TxIN26	30	RxOUT26	6
VSYNC	TxIN25	28	RxOUT25	5
HSYNC	TxIN24	27	RxOUT24	3
RESERVED	TxIN23 (Note 1)	25	RxOUT23	2

Note 1: Connect TxINPUT23 to ground if unused.

TABLE 2. Recommended 6-Bit Color Mapping (for 48L Devices)

Color Bit	TX Input Name	TX Pin Number	RX Input Name	RX Pin Number
R0-LSB	TxIN0	44	RxOUT0	24
R1	TxIN1	45	RxOUT1	26
R2	TxIN2	47	RxOUT2	27
R3	TxIN3	48	RxOUT3	29
R4	TxIN4	1	RxOUT4	30
R5-MSB	TxIN5	3	RxOUT5	31
G0-LSB	TxIN6	4	RxOUT6	33
G1	TxIN7	6	RxOUT7	34
G2	TxIN8	7	RxOUT8	35
G3	TxIN9	9	RxOUT9	37
G4	TxIN10	10	RxOUT10	39
G5-MSB	TxIN11	12	RxOUT11	40
B0-LSB	TxIN12	13	RxOUT12	41
B1	TxIN13	15	RxOUT13	43
B2	TxIN14	16	RxOUT14	45
B3	TxIN15	18	RxOUT15	46
B4	TxIN16	19	RxOUT16	47
B5-MSB	TxIN17	20	RxOUT17	1
ENABL	TxIN20	25	RxOUT20	5
VSYNC	TxIN19	23	RxOUT19	4
HSYNC	TxIN18	22	RxOUT18	2



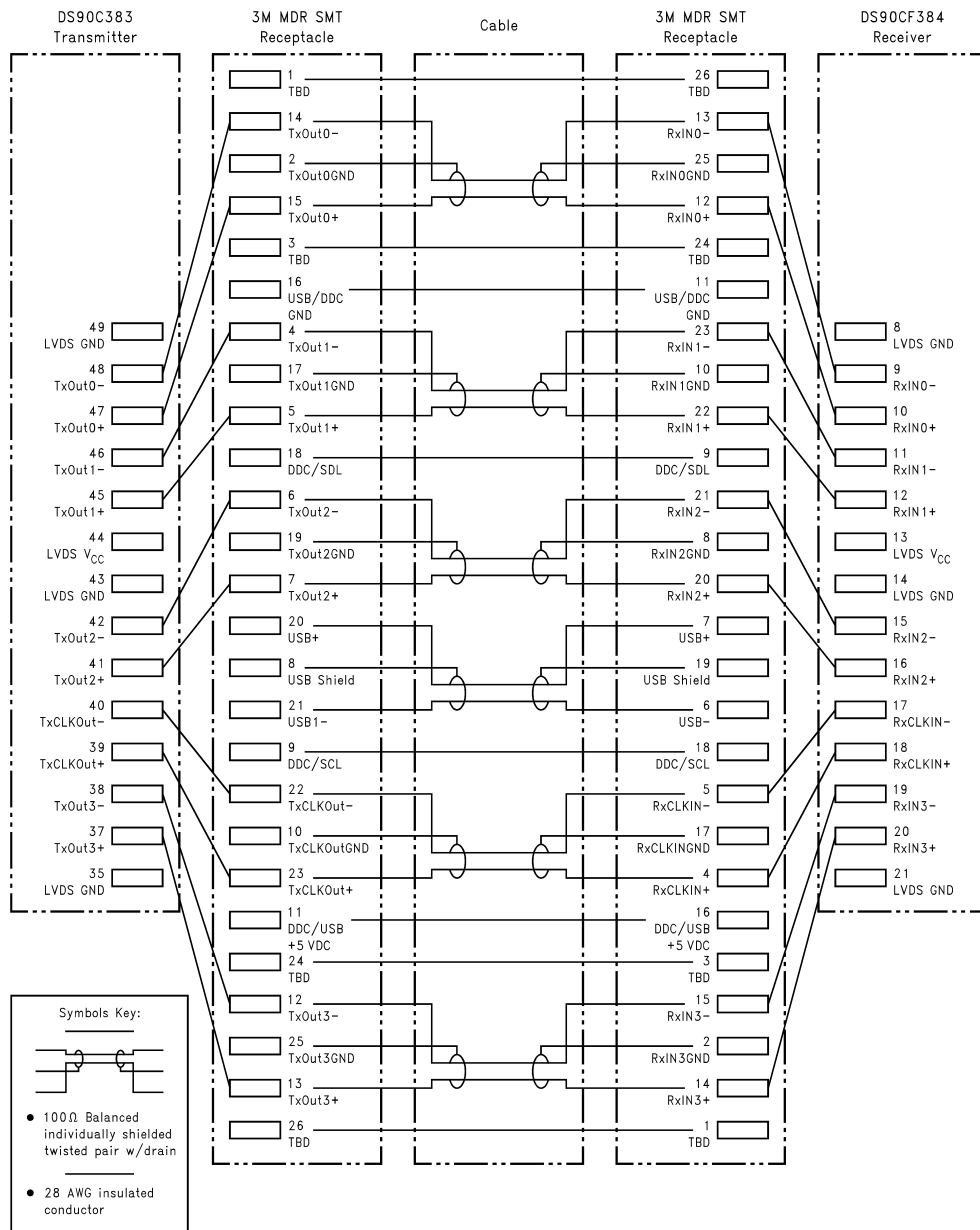
AN100098-6

FIGURE 6. Recommended Color Bit Mapping

RECOMMENDED CONNECTOR PINOUT

For monitor applications where the system (PC) is provided by one manufacturer and the Flat Panel Display is provided by another a standardized connector (mechanical specifications and pinout) is required along with the standardized color bit mapping to ensure inter-operation. As noted above the 3M MDR High Speed Differential Transmission system is an ideal cable for this application from a skew, impedance, and connector point of view for long cables (10 meter). The 26 position connector is becoming popular for use with a

FPD-Link digital interface supporting 6-bit or 8-bit color. Enough contacts are provided to also support DDC/SDL and USB along with power and ground lines and a few spares. This pinout is referred to in the 3M documentation as the F26-7 pinout for use with version 9.3 MDR cable (p/n 14526-EZ8B-XXX-07C Series). This pinout is shown in *Figure 7*. In addition a standardization effort is under way to specify a 28 position connector and a variety of cable media (standard twisted pair cables to twin-ax cables - application dependent).



AN100098-7

FIGURE 7. 26-Pin MDR Pinout

SUMMARY

High speed PCB and Interconnect design practices should be employed to ensure an error free low emission design for the FPD-Link 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 of the RX devices. Matching impedances within 10% is recommended to reduce the creation of reflections (even mode) along the interconnect/cable.

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.
- Traces should be designed for differential impedance control (space between traces needs to be controlled). See AN-905 for equations.
- 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 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 (V_{CC}, LVDS V_{CC}, and PLL V_{CC}).

APPENDIX

Related FPD-Link and LVDS Application Notes from National:

AN-XXX	Topic
AN-1032	Introduction to FPD-Link
AN-1035	PCB Design Recommendations
AN-1056	STN Panel Applications
AN-1059	SKEW and Jitter Calculations
AN-1084	Links in Parallel
AN-905	Differential Impedance Calculations

Related Application Notes

High speed LVDS Transmission Information:

High Performance Shielded Twin-ax Cable Assembly, Part Number 14526

SMT R/A Receptacle (26 position), Part Number 10226-1210VE

3M Tech Paper: "Selecting the "Right" Right Angle Interconnects: Surface Mount vs Through-hole" by Francis G. Hart
3M Electronics products Division, Austin, Texas; Kay Tohyama, Yamagata 3M Limited, Yamagata, Japan.

LIFE SUPPORT POLICY

NATIONAL'S PRODUCTS ARE NOT AUTHORIZED FOR USE AS CRITICAL COMPONENTS IN LIFE SUPPORT DEVICES OR SYSTEMS WITHOUT THE EXPRESS WRITTEN APPROVAL OF THE PRESIDENT AND GENERAL COUNSEL OF NATIONAL SEMICONDUCTOR CORPORATION. As used herein:

1. Life support devices or systems are devices or systems which, (a) are intended for surgical implant into the body, or (b) support or sustain life, and whose failure to perform when properly used in accordance with instructions for use provided in the labeling, can be reasonably expected to result in a significant injury to the user.
2. A critical component is any component of a life support device or system whose failure to perform can be reasonably expected to cause the failure of the life support device or system, or to affect its safety or effectiveness.



National Semiconductor Corporation
Americas
Tel: 1-800-272-9959
Fax: 1-800-737-7018
Email: support@nsc.com

National Semiconductor Europe
Fax: +49 (0) 1 80-530 85 86
Email: europe.support@nsc.com
Deutsch Tel: +49 (0) 1 80-530 85 85
English Tel: +49 (0) 1 80-532 78 32
Français Tel: +49 (0) 1 80-532 93 58
Italiano Tel: +49 (0) 1 80-534 16 80

National Semiconductor Asia Pacific Customer Response Group
Tel: 65-2544466
Fax: 65-2504466
Email: sea.support@nsc.com

National Semiconductor Japan Ltd.
Tel: 81-3-5639-7560
Fax: 81-3-5639-7507

www.national.com