PCB Design Guidelines for LVDS Technology

National Semiconductor
Application Note 1035
Syed B. Huq
August 1996

INTRODUCTION

Technology advances has generated devices operating at clock speeds exceeding 100 MHz. With higher clock rates and pico seconds edge rate devices, PCB interconnects act as transmission lines and should be treated as such. Reflections due to mismatched impedance, cross-talk, die-electric loss, skin effects, dispersion loss and reduction of noise margin are some of the undesirable events seen on high-speed transmission lines. At the same time, the bandwidth limiting factors of the interconnect must be understood. It is these effects that greatly reduce the performance of Si once placed on a PCB.

Understanding controlled impedance, differential signaling layout, de-coupling, terminations, layer stack-up and stub effects, can minimize many pitfalls and reduce cycle time in designing PCB's. This application note gives the PCB designer some common guidelines to follow in designing PCB's for LVDS (Low Voltage Differential Signaling) technology.

CHOOSING THE PROPER MATERIAL FOR PCB

Proper selection of material for high-speed board is essential. As the signal propagates through the interconnect, due to the lossy nature of the interconnect, signal degradation occurs. Er (dielectric constant) of the material and loss tangent or tan delta (Tanδ) are some of the key parameters that explain the lossy nature of PCB's. Er relates to a material's capability to hold charge and Tanδ relates to how much of the energy is lost in the material due to dissipation. Ideally, materials should be selected with the lowest Er and Tanδ. Table I below shows some typical numbers:

TABLE I

Material Selection	Dielectric Constant (Er)	Loss Tangent (Tanδ)
Air	1.0	0
PTFE (Teflon)	2.1–2.5	0.0002–0.002
BT Resin	2.9–3.9	0.003–0.012
Polyimide	2.8–3.5	0.004–0.02
Silica (Quartz)	3.8–4.2	0.0006–0.005
Polyimide/Glass	3.8–4.5	0.003–0.01
Epoxy/Glass (FR-4)	4.1–5.3	0.002–0.02

(Source: DesignSuperCon '95)

As the table above shows, Er is not a constant as we know it and can vary considerably. In most applications, FR-4 material is used, FR-4 material has acceptable performance up to the 100 MHz range, beyond that Teflon should be considered. Teflon tends to be 4X more expensive than FR-4 but as Table I shows, Teflon has a lower Er thus low loss. The lower the Er, the faster the velocity of propagation (Equation 1), the faster the board. The key point to remember is to minimize signal degradation as much as possible.

$$\sqrt{Er} = \frac{C}{V} \qquad \text{Equation (1)}$$

Where, C = 0.0118 in/ps (Speed of light)
V = Velocity of Propagation
Er = die-electric constant

PCB manufacturers publish a datasheet along with their boards. This datasheet specifies the Er and Tanδ along with other electrical parameters. Due to process variation, both the Er and Tanδ can change from board to board. Ensure that your PCB manufacturer provides this data on the fabrication material of your board.

You can think of fabricating a PCB board as laying a carpet. The glass-fiber material is held together by epoxy material. Places where the weave is not tight is where the Er can change. Within a single FR-4 PCB a 10% variation of Er is not uncommon and this can alter the propagation velocity of the signal considerably. This can lead to skew issues on signal lines.

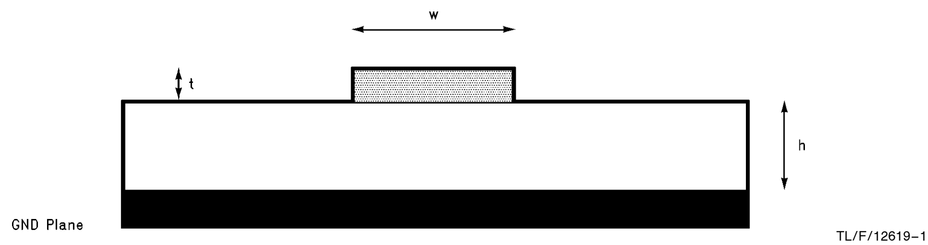
MICROSTRIP OR STRIPLINE?

Microstrip is a PCB trace above the dielectric material whereas a strip line is embedded between the substrate. The decision to use either one mainly depends on the number of layer the boards will be constructed and the complexity of signal routing. Some designs use a combination of both. Usually boards are either 4 layer or 6 and in some cases 8. Due to the high-speed nature of LVDS lines, they need to be seperated from larger swing TTL lines to avoid cross-talk. This can be done by using signals on different layers in a PCB isolated by Power/GND planes.

Delay through a stripline is longer than that of microstrip. Typically, microstrip has a delay of 147 ps/in (FR-4) and stripline has a delay of 188 ps/in (FR-4). Striplines require more via's than microstrip.

HOW TO CALCULATE FOR CONTROLLED IMPEDANCE (SINGLE-ENDED)

For a micro-strip (see *Figure 1*).



$$Z = \frac{87}{\sqrt{Er + 1.41}} \ln \left(\frac{5.98h}{0.8w + t} \right)$$

Where: Z = trace impedance

Er = die-electric constant

Er = 4.5 (FR-4)

Er = 2.4 (Teflon)

w = width of trace

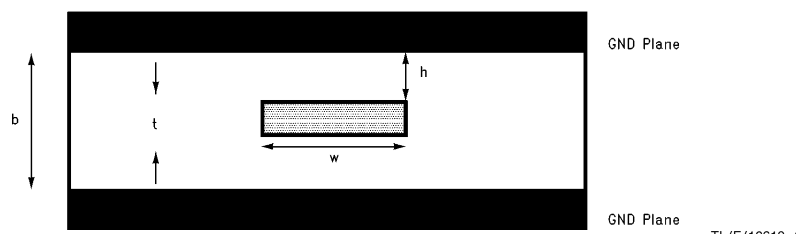
t = thickness of trace

h = prepeg (die-electric) height

FIGURE 1. Microstrip

Suggested typical numbers for LVDS PCB using microstrip are, t = 1.4 mils, w = 12.0 mils and h = 8.1 mils. This will give a 50Ω impedance from each trace to ground. The critical dimension that must also be taken into account is the distance between the signal pairs. This controls the differential impedance. It is recommended to hold this separation distance constant as much as possible (minor violations may occur at the device and connector connections). Next calculate the resulting differential impedance, and check that it matches the selected media (cable) differential mode characteristic impedance. If it is off substantially, the PCB trace dimensions should be adjusted to provide a match. To prevent reflections the PCB trace pair's impedance should be matched to the interconnect/media. This is also the ideal value for the termination resistor that is connected across the pair at the receiver's input. See AN-905 for more details on the calculation of differential impedance.

For a strip line (see *Figure 2*).



$$Z = \frac{60}{\sqrt{Er}} \ln \left(\frac{4b}{0.67\,\pi\,(0.8\,w + t)} \right)$$

Where: Z = trace impedance

Er = die-electric constant

Er = 4.5 (FR-4)

Er = 2.4 (Teflon)

w = width of trace

t = thickness of trace

b = height between GND planes

h = prepeg (die-electric) height

FIGURE 2. Stripline

ROUTING OF DIFFERENTIAL LINES

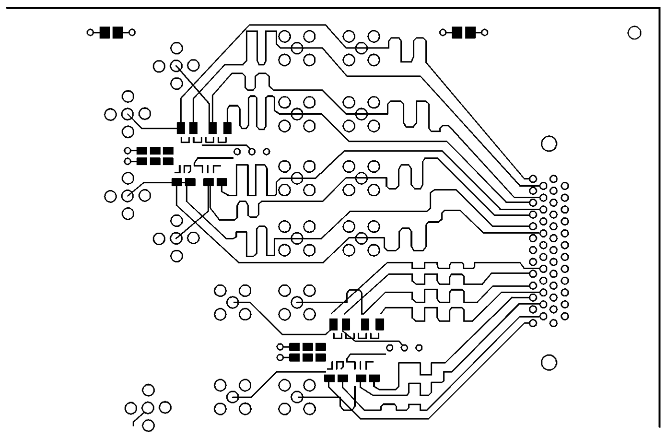
Differential lines have the characteristics of cancelling common mode noise as it appears in phase on the two lines. This cancellation of common mode noise and magnetic effects allow differential drivers and receivers to operate over longer distances. But, this will only work if the two traces are running close to each other. Care should be taken to keep the differential lines as parallel as possible. It is also critical to maintain the electrical/physical length of the two traces to be identical. This guarantees no skew on the line.

A sample layout of LVDS traces *(Figure 3)* shows the mitering effects done on the differential lines to match the electrical/physical lengths.

A sharp orthogonal turn (90°) should be avoided as it causes a sharp change in impedance. An arc should be used instead to round off the edge. This is critical in high frequency board layout.

CROSS-TALK BETWEEN TTL AND LVDS LINES

The PCB designer must note that all TTL/CMOS signal paths need to be isolated from the LVDS signal lines. Crosstalk is directly proportional to dv/dt. Since TTL/CMOS lines have a larger swing, crosstalk can easily occur if the TTL/CMOS paths are right next to the LVDS lines. Separation of the two technologies needs to be made either by increasing the distance between the two or running a ground trace between the two or isolating by using different planes.



TL/F/12619–3

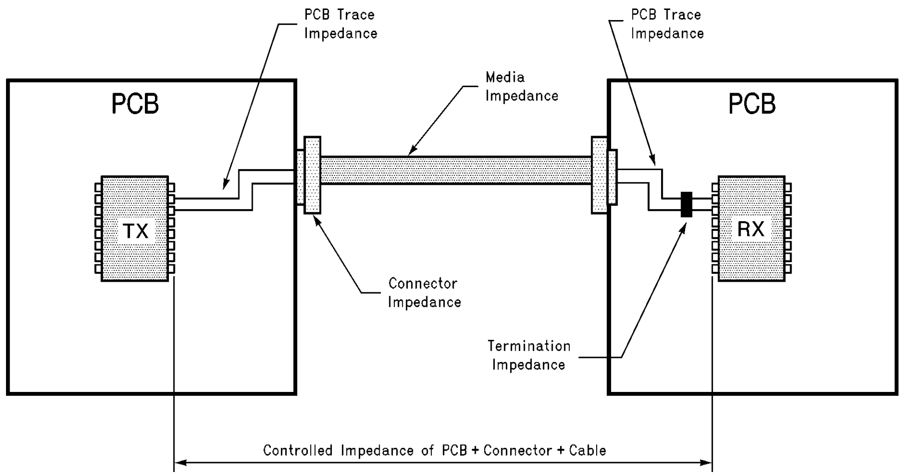
FIGURE 3. Trace Mitering on LVDS Lines

When configuring a connector interface to a PCB and also for cable/connector interface from PCB to PCB, the designer should allow GND pins in between signal pairs on a connector as well as in a cable.

MISMATCHED IMPEDANCE

A PCB trace can be between 50Ω to 110Ω. As shown in *Figures 1* and *2,* this is dependent on the width and distance from the trace to ground. If a trace is wide and close to the ground plane, it is more capacitive and has an impedance close to 50Ω. If the trace is narrow and a good distance from a ground plane, it is more inductive and approaches 110Ω. Controlling the impedance to the desired value is very critical in avoiding reflections on the line.

Matching of impedance to reduce reflection should also be considered *(Figure 4).* Mismatch in impedance cuts into the noise margin and can ultimately render an application useless. The idea is to properly match the impedance of the media being driven with the same value of termination resistor.

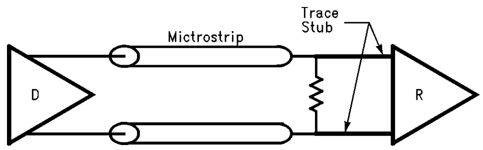


TL/F/12619–4

FIGURE 4. Matching Impedance

TERMINATION RESISTOR PLACEMENT AND STUBS

LVDS Drivers are current mode and they require the termination at the far end of the cable (PCB trace) and as close to the Receiver's inputs as possible. An inch of PC trace stub (see *Figure 5*) can act as an unterminated transmission line causing reflection and this stub should be minimized. Multiple reflections causes ringing, overshoot and undershoot which reduces the noise margin.



TL/F/12619–6

FIGURE 5. Termination and Trace Stub

In case of probing an LVDS tranmission line, a high impedance (> 1 MΩ) scope probe should be used with a high bandwidth (BW) in the range of 3 GHz–5 GHz and low capacitance (0.25 pF–0.5 pF). Improper probing of a high speed transmission line can give deceiving results.

CHOOSING TERMINATION RESISTORS (SMT)

Various vendors offer resistors in SMT form. A terminating resistor with radial shape can have too much inductance and this should be avoided. Only SMT type chip resistors with a tight tolerance should be used. Recommended tolerance is ± 1% of the termination value (100Ω typical). The user should always use $R_t = Z_O$, termination resistor matched to the media. LVDS Drivers require a termination resistor with a range of 90Ω to 120Ω. SMT form factor also helps to reduce EMI.

DECOUPLING OF V_{CC} LINE

Both the main supply line and the device V_{CC} pins should be properly decoupled. Bulk decoupling at the supply provides a constant low amplitude longer duration current and localized decoupling at the device provides high frequency energy required during SOS (simultaneous output switching) events. Proper decoupling reduces voltage spikes when all I/O pins are simultaneously switching. When decoupling the device V_{CC} pins, the capacitors need to be placed as close to the device V_{CC} pin as possible. Inductance caused by longer lead lengths must be avoided.

MLC (Multi-Layered Ceramic) capacitors in SMT form factor are recommended. Under high frequency, a capacitor is not an ideal element and can behave more like a LCR element. The L and the R are parasitic effects that cause more problems. Since the goal is to reduce inductance in the line, short leads or SMT form factor is critical. It has been seen that a 2:1 length-width aspect ratio can have 2 nH of inductance but a 1:2 (width exceeds length) aspect ratio can cut down the inductance to 0.5 nH in MLC capacitors. For the FPD Link chip set, the recommended bypassing is as follows:

Power Supply: 10.0 μF tantalum electrolytic capacitor

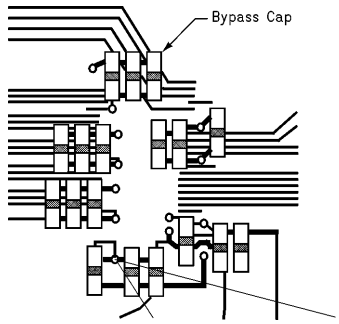
V_{CC}: 0.1 μF, 0.01 μF and 0.001 μF

LVDS V_{CC}: 0.1 μF, 0.01 μF and 0.001 μF

PLL V_{CC}: 0.1 μF, 0.01 μF and 0.001 μF

A sample layout of FPD Link *(Figure 6)* with multiple decoupling per V_{CC} pin is illustrated below. Note the three MLC SMT capacitors per V_{CC} pin. V_{CC} and GND trace width should be wider to reduce inductance and multiple via's are also recommended to help reduce inductance.

In addition to low series inductance, lower effective series resistance (ESR) should be considered when choosing capacitors.



TL/F/12619–5

FIGURE 6. Bypass Cap Layout

SUMMARY

High frequency effects should be minimized as much as possible when designing a PCB. Effects such as cross-talk, mismatched impedance, die-electric loss, skin effects, stubs can cut into the noise margin of the design. By following the PCB design guides mentioned in this application note, a designer can ensure success in developing a PCB for LVDS technology by avoiding all the high speed pit falls that are present in PCB designs today.

REFERENCES

Designing the System from the Materials Up, Martin P. Goetz: DesignSupercon95.

LinesimPRO owners manual: Hyperlynx Inc.

LVDS Demonstration Board: National Semiconductor Corp.

AN-971: An Overview of LVDS Technology: National Semiconductor Corp.

National Semiconductor Corporation
1111 West Bardin Road
Arlington, TX 76017
Tel: 1(800) 272-9959
Fax: 1(800) 737-7018

http://www.national.com

National Semiconductor Europe
Fax: +49 (0) 180-530 85 86
Email: europe.support@nsc.com
Deutsch Tel: +49 (0) 180-530 85 85
English Tel: +49 (0) 180-532 78 32
Français Tel: +49 (0) 180-532 93 58
Italiano Tel: +49 (0) 180-534 16 80

National Semiconductor Hong Kong Ltd.
13th Floor, Straight Block,
Ocean Centre, 5 Canton Rd.
Tsimshatsui, Kowloon
Hong Kong
Tel: (852) 2737-1600
Fax: (852) 2736-9960

National Semiconductor Japan Ltd.
Tel: 81-043-299-2308
Fax: 81-043-299-2408