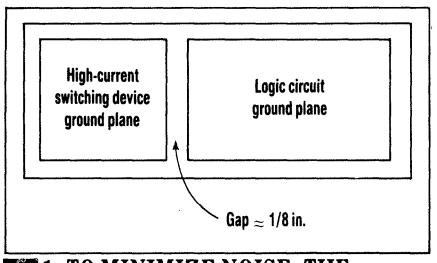
MAKE SURE THAT YOUR TURBO-CHARGED LOGIC SYSTEM WORKS BY PAYING AS MUCH ATTENTION TO PRINTED-CIRCUIT BOARD LAYOUT TECHNIQUES AS TO LOGIC DESIGN CONSIDERATIONS.

AVOID THE PITFALLS OF HIGH-SPEED LOGIC DESIGN

Modern high-speed systems demand modern high-speed logic families. Consequently, semiconductor houses have developed such product lines as ACT, FACT, and AS. But these systems also demand that the lay-out of their boards conform with the results of distributed-element theory, otherwise ringing, crosstalk, and other transmission-line phenomena render those systems inoperative. Meeting this second requirement necessitates something more than a new product introduction—it insists on a change in the way logic boards are engineered. The logic-systems designer and the board-layout designer must work hand-in-hand if a viable high-speed board or system is to be produced.

In the past, logic design and board layout were usually regarded as separate parts of the design process. First the system designer configured the logic, then the board engineer laid it out. That approach worked because slew rates were so low (0.3 to 0.5 V/ns) that crosstalk wasn't much of a problem; rise times were so long (4 to 6 ns) that ringing could settle down before a logic element could change state; and in general, the assumptions of lumped-element circuit theory usually worked out pretty well.

For systems designed with today's high-speed logic circuitry, those underlying assumptions no longer hold true. Today's slew rates are on the order of 2 to 3 V/ns, rise times are below 2 ns (frequently, below 1 ns), and transmission-line phenomena, such as ringing, can be a problem for trace



1. TO MINIMIZE NOISE, THE ground plane should be fragmented into separate areas for noisy high-current devices and for sensitive logic circuits. For best results, the number of signal lines that cross the gap between the fragments should be minimized.

JOCK TOMLINSON
Lattice Semiconductor Corp., P.O. Box 2500, Portland, OR 97208; (503) 681-0118.

Reprinted with permission from **ELECTRONIC DESIGN** - November 9, 1989

DESIGNING WITH HIGH-SPEED LOGIC

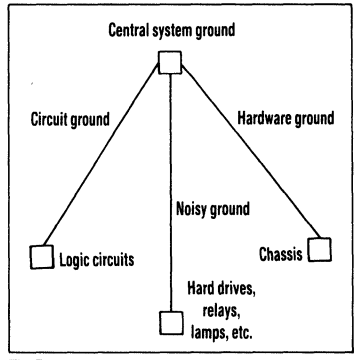
lengths as short as 7 in. As a result, logic designers must take certain steps:

- Use ground and power planes.
- Control conductor spacings to eliminate crosstalk.
- Make extensive use of decoupling capacitors.
- Pay attention to ac loading.
- Terminate lines properly to minimize reflections.

PLANE ADVICE

For high-speed logic, ground planes aren't simply suggested for reliable board performance—they are absolutely necessary. It's essential that one layer of the board be assigned for a ground plane and that it cover as large an area as possible. A solid ground plane lowers the ground-return-path impedance as well as the device-to-device ground pin impedance.

But a common ground plane for all of the circuitry in a system can cause problems by coupling noise from high-current switching devices into sensitive logic inputs. Therefore, the ground plane for such high-current devices as relays, lamps, motors, and hard drives should be separated from the logic ground. This can be accomplished by fragmenting the ground plane into discrete areas *(Fig. 1)*.



2. SEPARATE DEDICATED grounds should be supplied for the logic circuitry, noisy high-current devices, and the chassis. The three should come together at one point, the central system ground, which is usually located near the power supply.

But fragmentation causes problems of its own—it creates discontinuities in the characteristic impedance of any transmission line that crosses the separation between fragments. Therefore, for best results, boards should be laid out so that only two fragments are needed. The gap between those fragments should be kept as narrow as possible (an eighth of an inch works well in most applications), and the number of signal lines that cross the gap should be minimized. Designers should also bear in mind that through-holes and vias subtract from the effective area of the plane, increasing its effective impedance.

As with grounding, an entire layer of the board should be designated as a power plane. Even though it is at a different potential, the power plane should be implemented in accordance with the same concepts as the ground plane. Therefore, it should be fragmented when necessary to isolate noisy components from delicate logic circuits.

A WELL-GROUNDED SYSTEM

In addition to properly designed power and ground planes, high-speed logic systems require the establishment of a good, clean (low-

SIGNAL LINES BECOME TRANSMISSION LINES

For the transmission line model illustrated in the diagram, the rise time (t_R) is less than the line propagation delay (T_D). In other words, a complete TTL level transition will occur before the pulse is received at the receiving end of the line and reflections (ringing) will result. The voltage change at point A on the line is expressed in Eq. 1:

$\Delta V_A = \Delta V_{int} (Z_O / (R_O + Z_O))$

Where: V_{int} = internal voltage on the output of the driver;

R_O = output impedance of the driving gate;

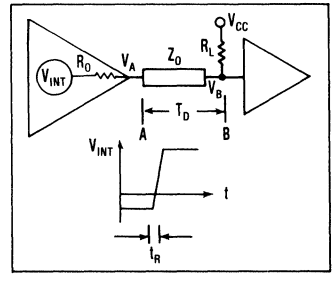
R_L = load impedance;

Z_O = the characteristic line impedance;

and V_A = the source voltage at the sending end of the line.

Because R_O is so small when compared to the line impedance, the change in voltage at point A (ΔV_A) will approximately equal the change in internal voltage (ΔV_{int}). This voltage transition propagates down the line and is seen at point B after the line propagation delay, T_D.

At point B, a portion of the wave will be reflected back towards point A in accordance with



the formula (Eq. 2):

Eq. 2

$\rho_L = (R_L - Z_O) / (R_L + Z_O)$

where ρ_L, called the voltage reflection coefficient (rho), is the ratio of the reflected voltage to the incident voltage.

After examining Eq. 2, it should be evident that $-1 \leq \rho \leq +1$. It should also be evident that there will be no reflected wave if $R_L = Z_O$—if the line is terminated in its characteristic impedance. Note that the reflected wave can, in principle, be as large as the incident voltage and of either positive or negative polarity.

This analysis holds true for the sending end of the line, as well as the receiving end. That is,

Eq. 3

$\rho_S = (R_O - Z_O) / (R_O + Z_O)$

DESIGNING WITH HIGH-SPEED LOGIC

noise) system ground for reliable performance. A clean system ground ensures less noise within the system, and thus ensures good, strong transistor margins. At least 10% of the ground connections on the pc card should be connected to the system ground to reduce card-to-ground impedance.

Like the ground and power planes of the individual boards, the overall grounding scheme should be fragmented with separate conductors provided for the various sections of the system. For example, all relays, lamps, hard drives, and other noise-generating devices should have their own separate ground path. The system's mechanical package (chassis, panels, and cabinet doors) should have a dedicated ground. And, of course, the logic circuitry should have a ground of its own.

Those three grounds should then come together at the central system ground point, which will usually be located near the power supply *(Fig. 2)*. This common-point grounding technique can also be very effective in reducing radiated interference (EMI and RFI).

TAMING CROSSTALK

Crosstalk—the undesirable coupling of a signal on one conductor to one on a nearby conductor—becomes an increasingly serious problem as slew rates go up. This signal coupling is made worse if the second trace has a high impedance or if the traces run parallel to one another for more than a few inches and are spaced less than 100 to 150 mils apart.

Crosstalk can be catastrophic to a logic board, sabotaging a conceptually flawless piece of logic design. For example, if a clock line and a data line run parallel to each other for more than several inches, and if the data line cross-couples or superimposes its signal onto the clock line, the device that the clock is driving may detect an illegal level transition.

Methods to reduce crosstalk are straightforward, though not particularly elegant. The coupling can be attenuated by separating the adjacent traces as much as possible. The trouble with this approach is that available board real estate often limits the possible separation to an inadequate amount.

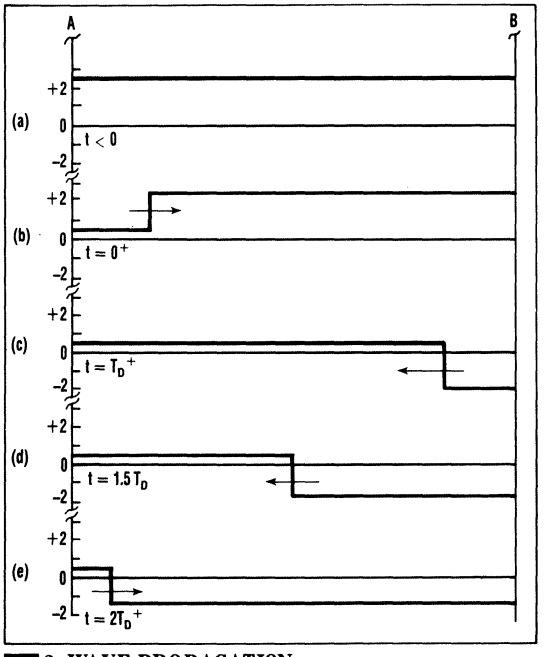
Ground striping, or shielding, is an effective way to reduce crosstalk and it makes better use of available board area. With ground striping, a ground trace (the stripe) is run between the two parallel traces to act as a shield. If ground striping is used, through holes to the ground plane should be placed every 1 to 1.5 inches along the ground strip to eliminate the possibility of inadvertently creating a stub or a high-frequency antenna.

Another step that can be taken to reduce crosstalk is to lower the impedance of those traces into which crosstalk is especially to be avoided. The lower the impedance that a trace presents, the harder it will be to cross-couple a signal into it.

Even with the use of power and ground planes on a pc board, decoupling capacitors must be used on the V_{CC} pins of every high-speed device. Those devices demand a nearly instantaneous change in current whenever they switch states. Because the power plane can't meet that demand, a high-quality decoupling capacitor is required, otherwise the switching will cause noise on the V_{CC} plane.

A 0.1-μF multilayer ceramic (MLC) or other RF quality (low-inductance) capacitor should be placed on every fast-slew-rate device as close to the V_{CC} pin as possible. The commercially available DIP sockets with built-in decoupling capacitors also work well in this application.

Most designers, when they think of loading at all, think in terms of dc loading—traditionally referred to as fan-out and fan-in. But that type of loading rarely presents a problem with today's state-of-the-art logic devices. Much more significant when designing with high-speed logic are input and output ac loading.

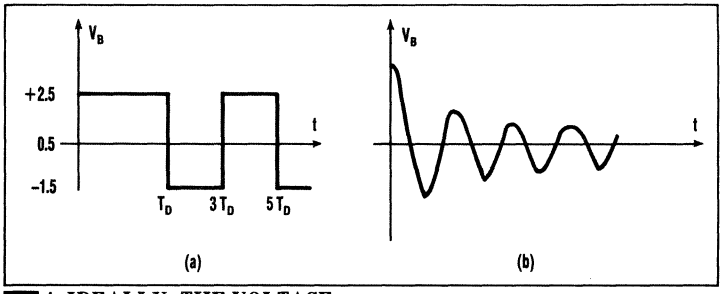


3. WAVE PROPAGATION along a transmission line occurs as follows: Prior to time zero, there is a steady-state voltage of 2.5 V dc on the line (a). At t = 0, the voltage at point A drops to 0.5 V, sending a negative pulse of -2 V toward point B (b). At t = T_D, that negative pulse is reflected from point B. It adds algebraically to the 0.5 V on the line and sends a -1.5-V pulse back toward point A (c). The reflections then continue as in (d) and (e).

INPUT CAPACITANCE

Because the input capacitance of a device impacts the overall performance of the logic circuit, it should be examined before a particular device is selected for a design. To ensure specified performance, the total load capacitance that a device drives—including the distributed ca-

DESIGNING WITH HIGH-SPEED LOGIC



4. IDEALLY, THE VOLTAGE at point B oscillates forever between +2.5 V and -1.5 V (a). In reality, it will be a damped ringing (b).

pacitance of the trace—shouldn't exceed the device's specified capacitive load. Most high-speed logic devices have a maximum loading of 50 pF. As a rule of thumb, the maximum load on any logic element should be no more than four to six devices for best speed/load performance. However, there are some high-slew-rate devices on the market that have higher output drive capabilities.

Beware Of Autorouter

The most common reason for not following the board-layout principles mentioned so far is having an autorouter do the layout. Autorouters do what they were designed to do very well: They place traces so as to make the most efficient use of the pc-board real estate. But most autorouters don't have the capability to determine which devices are high-speed and which are not. This is where the logic designer must step in and lay out sections, or islands, of high-speed logic by hand in order to avoid the pitfalls of designing with high-speed logic.

Transmission Lines

In addition to the common-sense layout considerations discussed so far, designers of high-speed systems must have at least a basic understanding of transmission lines and proper termination techniques *(see "Signal Lines Become Transmission Lines," p. 76)*. The reason: As frequencies go up, wavelengths come down to the point where they are of the same order as circuit-board dimensions. Once that happens, any connection between devices should be considered a transmission line. The lumped-element assumption is simply invalid above that point.

The most common consequence of failing to consider the distributed nature of a high-speed logic board is ringing, which is caused by multiple reflections from the ends of unterminated transmission lines. An unterminated line has no load impedance ($R_L = \infty$) and is therefore an impedance-mismatched line. The behavior of this line when connected to a device with a fast slew rate can be understood from the following example: Prior to time zero, there's a steady-state voltage of 2.5 V dc at all points on the line *(Fig. 3a)*. At $t = 0$, an initial TTL voltage transition from 2.5 V to 0.5 V occurs at point A *(Fig. 3b)*. Time T_D later, the signal reaches point B and is reflected by the load reflection coefficient, ρ_L.

The input impedance of the device at point B is very high with respect to Z_0; R_L can be approximated by infinity. By plugging into Eq. 2 from the box *(p. 76)*, the reflection coefficient approximately equals +1. In other words, the voltage reflected by the load is equal to the incident voltage *(Fig. 3c)*. The reflected wave passes back along the signal path toward point A *(Fig. 3d)*.

Repeating the calculations for the sending end of the line (point A), where $R_0 \simeq 0$, you get a value for the source reflection coefficient, ρ_S, of -1. In other words, there are reflections from the source as well as the load,, but the source reflects the inversion of the wave that is incident upon it *(Fig. 3e)*.

Looking just at the behavior of the signal at point B, the single-step volt-

RULES TO REMEMBER

The following ten rules summarize everything the logic designer needs to know when designing with high-speed CMOS.

1) Keep signal interconnections as short as possible.

2) Use a multilayer PCB.

3) Provide ground and power planes. Discontinuities in the planes should be avoided because reflections can occur from abrupt changes in the characteristic impedance.

4) Fragment the ground and power planes to supply separate sections for high-current switching devices.

5) Use decoupling capacitors on every high-speed logic device (0.1 μF MLC type) located as close to the V_{CC} pin as possible.

6) Provide the maximum possible spacing among all high-speed parallel signal leads.

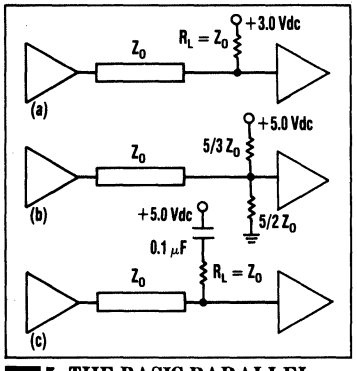
7) Terminate high-speed signal lines where $t_R < 2T_D$.

8) Beware of ac loading conditions within the design. Exceeding the manufacturer's recommended operating conditions, especially for capacitance, can cause problems.

9) When using parallel termination, put bends in all high-speed signal runs that go to more than one load. Use a termination load at the absolute end of the line.

10) Create islands of high-speed devices on the pc board. This simplifies board layout and ropes-off the high-speed areas.

DESIGNING WITH HIGH-SPEED LOGIC



5. THE BASIC PARALLEL termination scheme works well but requires a separate 3-V supply (a). The Thevenin equivalent eliminates the need for a separate supply, but dissipates extra power from the regular 5-V supply (b). The use of a capacitor cuts dc dissipation altogether while supplying ac termination (c).

age transition at t = 0 leads to an endlessly oscillating signal with a total voltage swing of 4.0 V—twice the original level transition. The voltage doubling comes about because the voltage at point B is the sum of the incident and reflected waves at that point *(Fig. 4a)*. Actually, because of the non-ideal nature of a real circuit board (finite input and output impedances, losses in the transmission lines, and so forth), ρ_L will be less than +1, and ρ_S will be greater than −1. As a result, the reflections will become successively smaller, causing the familiar damped ringing condition *(Fig. 4b)*.

If the ringing amplitude is large enough, it can cause the receiving device to see an illegal level transition and possibly result in spurious logic states occupying the logic design. In some cases, the amplitude of the ringing can actually be large enough to damage the input of the receiving device.

TERMINATE YOUR TROUBLES

The way to eliminate ringing on a transmission line is to terminate the line in its characteristic impedance at either the sending or receiving end. The most common way to terminate a line is with a parallel termination at the receiving end *(Fig. 5)*.

In the configuration *(Fig. 5a)*, $R_L = Z_0$ and R_L is pulled up to 3 V dc. In principle, R_L could be tied to ground, but TTL-compatible devices could not then supply the necessary drive.

Solving for ρ_L *(Eq. 2)*, it can be seen that $\rho_L = 0$. Terminating a line in its characteristic impedance results in a reflection coefficient of zero, which means that there will be no reflections or distortions on the line. Other than the time delay, T_D, the line will act as if it were a dc circuit. It's important to note that even though devices or gates may be placed at any location on the line, the terminating resistor should be placed at the end of the line. In no case should the line be split like a Tee to feed several devices in parallel *(Fig. 6a)*. Instead, it should be serpentined to feed them sequentially *(Fig. 6b)*.

The 3-V power source shown *(Fig. 5a)* appears at first to be a major drawback, but R_L and the power supply can be expressed as a Thevenin equivalent running off the system power supply of 5 V dc *(Fig. 5b)*. This variant works well, but the designer should bear in mind that it dissipates additional power.

REDUCING DISSIPATION

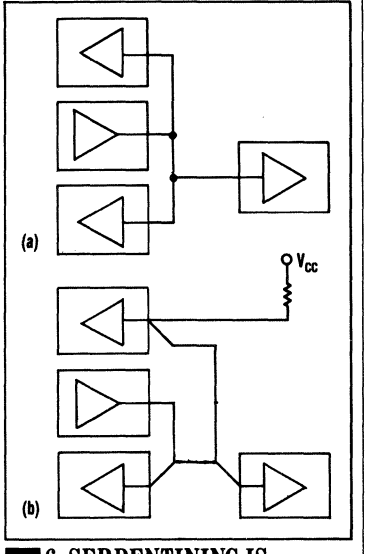
A solution that dissipates less power than either of the others uses a capacitor to cut the dc dissipation to zero *(Fig. 5c)*. The recommended capacitor is a 0.1-μF MLC type. Several manufacturers produce both capacitor-resistor and pull-up/pull-down termination packs. The pull-up/pull-down packs usually come in a single in-line package (SIP) with pins on 0.1-in. centers, while the capacitor-resistor combination comes in a standard 16-pin DIP. The most common SIP pull-up/pull-down resistor values are 220Ω/330Ω, 330Ω/470Ω combinations.

An alternative to a parallel termination at the receiving end is a series termination at the sending end *(Fig. 7)*. The idea behind serial termination is to make $\rho_S = 0$ and $\rho_L = +1$. To do so, R_L is made equal to infinity (left unterminated) and a series resistor is added at the source to make the overall source impedance equal to the characteristic impedance of the line—that is, $R_S + R_O = Z_{OL}$.

Making $R_S + R_O$ equal to Z_{OL}, of course, creates a voltage divider, which puts half of the signal amplitude across the line and half across the series combination of R_S and R_O. Therefore, with the series termination, the amplitude of the transmitted wave is half of what it would be without the termination.

Interestingly enough, the unterminated receiving end of the line precisely compensates for this halving of the amplitude. The reason is as follows: At the receiving end, the half-amplitude wave is received and a half-amplitude wave is reflected. But bear in mind that those are two separate waves whose amplitudes add at the point of reflection. As a result of this addition, the only thing seen at the receiving end of the line is a full-size pulse.

The main disadvantage of a series termination is that the receiving gate or gates must be at the end of the line—no distributed loading is possible. The obvious advantage of a series termination over a parallel one is that a series termination doesn't



6. SERPENTINING IS essential when terminating a line. Never split the line to feed parallel devices (a). Rather, feed them sequentially with a serpentined line (b).

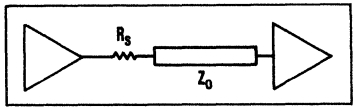
6

DESIGNING WITH HIGH-SPEED LOGIC

require any connection to a power supply.

Transmission-line effects must be taken into consideration whenever line propagation delays get up to the point where a signal transition can be completed before that signal can travel down a line, be reflected, and travel back to its starting point. In other words, lines must be terminated when,

$2T_D = T_R$.



7. THE SERIES termination needs no pull-up supply. Its main disadvantage is that it can't handle distributed loads.

CALCULATING DELAY

Taking 2 ns as a typical rise time for a state-of-the-art high-speed logic device, how long can a board trace get before its propagation delay gets to be 1-ns long? For a pc board with a continuous ground plane and a signal trace on the adjacent layer, the propagation delay depends on only one variable, the dielectric constant of the board material. That delay time is given by:

$$t_{PD} = 1.017\,(0.475\,e_R + 0.67)^{1/2}\ \text{ns/ft}$$

For a typical board constructed of FR4 material, e_R (the dielectric constant) is 4.7 to 4.9. If an average e_R of 4.8 is used in the equation, then t_{PD} turns out to be 1.75 ns/ft, which works out to 6.86 in./ns. As a rule of thumb, then, any line that is over 7 in. long should be considered a transmission line and approached accordingly.□

Jock Tomlinson, senior applications engineer at Lattice, holds a BSEE from Colorado State University.