FAIRCHILD
SEMICONDUCTOR™

March 2002
Revised April 2002

Section 8 - Mechanical Considerations
Backplane Designer's Guide

The application and performance requirements of a backplane will define its physical layout and size. When the minimum physical structure requirements are known, the form factor can be selected. Application and performance requirements also help the designer decide if the backplane will be a full custom design, a completely packaged protocol, or some point in between. Performance requirements will also define minimum requirements for chassis or cage, card selection, and backplane connectors.

The cage or chassis is the physical support system used to secure cards to the backplane. Connectors join the cards physically and electrically to the backplane.

The card size selected will determine the spacing between cards and overall size of the backplane. Therefore, card size has a direct effect on backplane performance.

The type of connector used can affect signal integrity because connectors have capacitance, increased rise time and flight time, increased noise (EMI and cross-talk), and effects on power consumption.

Section Reference

This section presents information covering backplane mechanical considerations:

- Layout and Form Factor
- Cage and Chassis Considerations
- Motherboards
- Card Dimensions
- Connector Considerations
- Cables and Cable Connectors
- Signal Routing

Section	Section Title	Contents
1	Introduction	Application demands, basic backplane considerations, and how to use this guide.
2	Backplane Protocols	Descriptions of different backplane bus protocols, including PCI- and VME-based protocols.
3	Backplane Architecture	Topics relevant to backplane configuration, including parallel versus serial configuration and different configuration topologies and timing architectures
4	Backplane Design Considerations	Issues relevant to backplane layout, including distributed capacitance, transmission line effect, stub length, termination, and throughput.
5	Backplane Signal Driving and Conditioning	Signal driving and conditioning, including power consumption, rise/fall time, propagation delay, flight time, device drive, pin conditioning, live insertion, and incident wave switching.
6	Noise, Cross-talk, Jitter, Skew and EMI	A review of the enemies of signal integrity and high frequency.
7	Transceiver Technologies	Detailed information about the following technologies: TTL-based (ABT, FCT, and LVT); ECL; and GTLP.
8	**Mechanical Considerations**	**Information about mechanical considerations such as backplane chassis/cages and connectors.**
9	Layout Considerations	Physical layout of the receiver and driver cards plugged into the backplane, primarily focusing on construction of the physical layer and the configuration of the devices that comprise the cards.

Backplane Layout and Form Factor

Simply defined, a backplane consists of a motherboard that has the primary function of transferring data. It contains connectors for daughter cards. Data processing is done in the daughter cards, and signals are routed through the backplane motherboard.

The majority of backplane motherboards are passive and contain only traces and connectors with little or no components on the motherboard itself. All active components on the daughter cards make repairs much quicker and easier. Also, daughter cards can be replaced quickly, and the defective card can be repaired without impacting system uptime. This is especially important in critical high-uptime systems such as telecommunications switching backplanes.

The exact implementation of a backplane including its form factor and electrical characteristics can be unique to the particular system. In some cases, a custom design is necessary to achieve performance and overall design requirements, particularly for very high-performance applications. The advantage of the custom approach is a backplane that closely matches system requirements, with the fewest possible compromises.

Figure 1 illustrates the layout of the Fairchild Semiconductor GTLP evaluation and demonstration backplane. This backplane uses a Compact PCI pinout but with customized signal and clock setup.

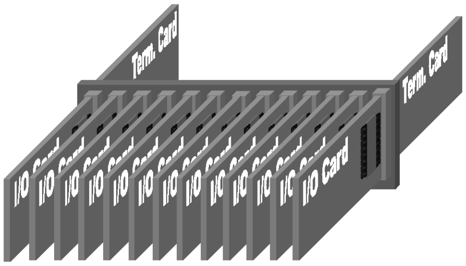

FIGURE 1. GTLP 21 slot multipoint incident wave backplane with backside termination cards. This backplane uses the Compact PCI pinout, but is a custom configuration.

Custom backplanes can be a design- and development-intensive process. And, with many of the parameters and components unique to the design, it can also be a costly approach.

An alternative to a unique custom backplane is a design that uses a backplane specification protocol that follows standards that vary in flexibility and performance. In most cases, these protocols are medium-performance multidrop designs that achieve high throughput by using wide bit widths. In some cases, these protocols are proprietary and must be licensed. Also, because the backplane will be designed to generic specifications, more compromises in final system performance may be required.

The advantages to using a specified protocol are cost and time savings. With the protocols already developed and specifications already set, much of the development work is done. Most of the components are readily available from vendors. These off-the-shelf components include almost everything required for backplane construction, including chassis, connectors, power supplies, motherboards, and, in some cases, daughter cards. More detailed information on some of the more widely used protocols can be found in Backplane Designer's Guide Section 2, "Backplane Protocols".

A third design approach is a semi custom or hybrid design. This is achieved by starting with one of the protocols and many of the available components, but designing for the speeds and layout required by the application. This design approach can save significant cost and time.

Because of the wide use of protocols and semi custom designs, this section will use some of the specified equipment to illustrate examples of various hardware components.

Backplane Chassis/Cages

The backplane enclosure is called a cage and also referred to as a chassis or rack. The implementation can be as simple as a chassis base and cover, or as sophisticated as a fully inclosed rack-mount system with card slides and integrated control panel. The form factor of the cage, its size, card size, and card spacing are based on system requirements or on the chosen protocol standard. The number of cards required in the backplane is another factor that determines the chassis to be used.

Backplane cages can be purchased in different sizes. The most popular sizes are 3U, 6U, and 9U. Other cages sizes are available including 4U and 12U. The U is an Electronics Industry Association (EIA) standard unit of measure equal to 1.75 inches (44.45mm) for equipment racks. Here are some examples:

- The 3U backplane cage is 133.35mm HIGH and accepts only 3U cards. These cards are 100mm HIGH with a typical 160mm Depth.
- The 6U backplane cage is 266.7mm HIGH and is frequently used in today's backplane applications because its size represents a balance between the 3U and the 9U. The standard 6U card is 233.35mm HIGH by 160mm Deep. An added feature of the 6U cage is that it can handle two different sized cards: the 6U and the 3U. This is important for the designer who has the requirement of a 3U card in the same backplane as a 6U card, although this type of application is not commonly used in backplane systems.
- The 9U backplane is typically used when dealing with larger-sized cards, and offers the potential for a higher amount of data transmission through the backplane.

Motherboard Configurations

Motherboard configurations and form factors follow system requirements, chosen protocol or architecture, and chassis or cage configuration. Some defined protocol standards have very narrow specifications, others have a wide degree of flexibility. If a protocol standard is chosen for the design implementation, the specifications may define the motherboard exactly. In many cases, vendors have motherboards that fit protocol specifications.

Whether a custom design or a protocol is chosen, there may be a wide latitude in configuration. These architecture choices have a significant impact on the end size and form factor of the backplane.

The physical layout of the backplane motherboard is a result of architecture choice. This will define the number of signal and power traces as well as how these traces are laid out on the board. The number and spacing of the daughter card connectors will also depend upon architecture and system requirements.

Signal and clock distribution design will have the most significant effect on the number and layout of motherboard traces. The most defining layout parameter is whether the backplane is a point-to-point or multidrop design.

In a point-to-point design, a trace will run only between two points. Usually this is implemented as a high-speed serial data stream. In some point-to-point applications, multiple channels are used. Point-to-point designs have many fewer trace runs than equivalent throughput multidrop systems.

A multidrop or multipoint backplane contains traces that connect multiple I/Os across the backplane. Multipoint is more common than multidrop, because it allows all I/Os both to receive and send data. The common implementation for this type of distribution is a parallel architecture that contains multiple data channels. High throughput is achieved with wide bit widths.

Further multipoint signal interconnection options include the standard end-to-end layout and the star distribution. End-to-end signal traces run from the first I/O to the last I/O on the backplane, with a connection to each I/O in between. The star interconnection routes all signals to the middle slot. The end-to-end interconnect uses less trace runs, but may require longer flight times. The star interconnect requires a more complex implementation, but has the potential for shorter flight times.

The clock distribution architecture also has an effect on form factor and complexity. Synchronous clocking architecture is a common implementation. A single clock source provides timing for all components requiring a clock. This is achieved by laying out clock lines of equal length to each device that requires a clock. The benefit is a single clock tree that sets system timing. However, maximum system speed is limited by clock flight time to the farthest component.

Source synchronous, in which each driving device clocks each of its receivers, has the potential to improve backplane performance. However, this type of clocking increases complexity because it requires many additional clock lines and received signal re-timing.

Asynchronous timing eliminates the clock and, therefore, many traces; however, it can require complex devices and software overhead.

More information on the architecture choices available is in Backplane Designer's Guide Section 3, “Backplane Architecture”.

Daughter Cards

As with motherboards, daughter card configurations and form factors follow system requirements, chosen protocol or architecture, and chassis or cage configuration. However, unlike motherboards, daughter cards are active components usually with different layouts and functions.

Whether designing a custom backplane or using a protocol standard, the designer usually has more latitude with daughter card layout. If a defined protocol is chosen for the design, many protocol-compatible functions can be purchased from vendors, reducing design and development costs.

As noted, some backplane cage sizes, such a 6U, can accept multiple card sizes. Coupled with protocol software that permits flexible card placement, this allows system modification and upgrades.

Power Supplies

Power supplies are available for all standard backplane card cages. A wide range of voltage and power options is available from multiple vendors. If a standard-size card cage is chosen for a design, a power supply for most power and voltage requirements can be purchased. This eliminates the cost of a custom designed power supply.

Backplane Connectors

The connector is the physical structure that joins the card to the backplane. In most backplane designs, the connector serves two functions: to physically connect and hold the daughter card on the backplane and to make the electrical connections necessary for operation.

Several different types of connectors are available. Selecting the best connector is important for good signal integrity, high throughput, and efficient system performance. With some protocols, the connector type and specifications are very strictly defined.

The size and type of backplane or the protocol often determines the size and type of the connector. The length of the connector often directly correlates to whether a 3U, 6U, or 9U-backplane cage is used. However, this may vary depending on design requirements and protocol. Some backplanes and daughter cards may be laid out for multiple connectors, allowing connector space for future expansion.

Connector Electrical Considerations

As noted, connectors are both a mechanical and an electrical interface. Due to this, the connector choice can affect the physical integrity and the electrical performance of the backplane.

Electrical parameters that affect the connector include system noise, capacitance, inductance, and power consumption. Depending on the application and the connector design, the connector can contribute to backplane noise characteristics, rise time, and flight time. Because of this, the connector choice has a direct effect on overall system performance.

To the signals running on the backplane the connector is seen as a stub. All the factors that affect the stub also affect the connector. Like the stub, a connector has capacitance and impedance when driven with high-frequency signals. If not controlled, this can disrupt the rise and flight times of signals on the backplane. As backplane frequencies increase, low connector capacitance and the impedance match of connector to stub and backplane becomes critical.

Another concern to the designer is noise. When operating at high frequencies, a backplane can create a significant level of noise, attributed to cross-talk, EMI, and ground bounce.

Connectors shield against and minimize noise with two standard methods. One connector design places metal shielding around the plastic structure of the connector. This metal structure will contain the internal noise of the connector and prevent any outside noise from disrupting the internal signals on the connector. There are many connectors on the market that incorporate shielding to minimize noise.

Another method of connector layout and design is to dedicate entire connector rows to ground to act as isolation. This helps isolate and shield connector signals. Grounded connector rows are required for some backplane protocols.

Nanosecond Pin Bounce

Nanosecond pin bounce, also called nano bounce or pin bounce, refers to oscillations that result when two metal components make contact. As they make contact there will be slight vibrations between the two conductors. This means that the metal contact will connect and disconnect for a number of occurrences over a time interval of only a few nanoseconds. When the two metal components connect they conduct and stop conducting when the connection is lost.

In many backplane applications systems must run 24 hours a day and cannot be shut down when a card needs to be inserted into, or extracted from, the backplane. Because of this, most backplanes are designed for live insertion. When a card is initially plugged into a live backplane, additional open and close events can be seen on the signal lines. These events (nanosecond pin bounce) occur in only a few nanoseconds.

Nanosecond pin bounce at the system level is seen as noise. In a high-performance backplane, the slight noise created can cause data corruption in the system.

This data was taken from RTC Magazine, September 2000. The data depicts an actual operating system during a hot-swap event. Notice the large positive disturbance that is created on the bus as the card is plugged into the backplane. (Courtesy of Motorola Computer Group)

One method of avoiding data corruption is to suspend bus data transfer during hot-swap. Many protocols allow for this. If bus suspension is not feasible, connectors with high-resistance ends are available from several vendors. These special connectors ensure that low resistance contact is not made until the card is fully seated.

Connector Mechanical Considerations

Connectors have many available mechanical feature options. With some protocols, the connector type and features are specified. When a particular connector is not specified, the layout and features chosen should reflect the requirements of the backplane design.

Keying, guide lugs, and polarizing lugs all have similar functions - safeguarding the equipment. Each of these features protects against incorrect insertion of boards as well as against different types of insertion faults.

Keying is the use of matched posts and sockets in the connector assembly of daughter cards and motherboards to prevent card insertion into incorrect slots. This is commonly employed to prevent over voltage damage to a card when multiple voltage levels are used in a system.

Guide lugs prevent the misalignment of connectors during the mating process. This prevents pins from being bent and ensures correct electrical contact. Polarizing lugs, as the name implies, ensure that cards cannot be inserted backwards.

Shrouds are connector surrounds that incorporate all the functions of keying, guides, and polarizing lugs without taking up pin space in the connector. They are available for some connector types.

Hot-plug connectors contain pins of differing heights. These pins allow the mating of ground first, followed by control and power signals before mating data signals, and they prevent glitches on the bus during live insertion. More information on live insertion is in Backplane Designer's Guide Section 3, “Backplane Signal Driving and Conditioning”.

Selection of a Connector

When selecting connectors for use in a backplane design, the following steps will help determine the minimum connector performance necessary for use in the backplane. Connector manufactures can supply specifications and information that will help ensure optimal connector selection.

1. Determine the amount of noise that will be seen by the connector by calculating the noise generation factor (NGF). NGF is a measure of the potential for a given combination of edge speed and voltage level to induce cross-talk and noise. The results take into account the signal-to-ground contact ratios and similar issues that affect the cross-talk of the connector. High NGF limits the number of connectors that meet the backplane's requirements. Low NGF increases the number of connectors that will meet the backplane's requirements.
2. If available, obtain the NGF specifications for connectors from manufacturers, although not all manufacturers provide NGF specifications.
3. Determine the signals per inch in the connector. The number of signals per inch will vary depending on the connector and the NGF. Different NGFs will affect the signal-to-ground ratio and therefore the number of pins available for signals.
4. Ensure the cross-talk values are within the signal integrity requirements of the system. The amount of cross-talk will vary with the level of the NGF.
5. Determine the card-to-card and pin spacing required.
6. Compare relative cost of the connector types available.

Routing Considerations

The choices made about how to route signals through connectors, cables, and boards in a system have many effects on the end product and specifically on system form factor, signal integrity, and system reliability.

As noted in the section on motherboards, the placement and number of traces will define the minimum board size required. Attempts to place traces inside acceptable minimum spacing has a negative effect on cross-talk and EMI. For this reason, minimum spacing and shielding requirements must be observed to ensure system signal integrity.

As frequency increases, spacing and shielding requirements increase and the use of shielded card connectors becomes mandatory.

Signal Routing Considerations

For best signal integrity and system reliability two basic layout rules should be followed:

- Be certain about the electrical compatibility of interconnects
- Isolate disparate signal types.

All interconnects on a signal line should have as close to the same resistance and impedance characteristics as possible. This includes traces, cables, connectors, and plated through-hole connections. Keeping the electrical characteristics the same minimizes signal discontinuities and noise from cross-talk and EMI.

Keeping like signal types together and isolated from other signal types reduces interference from cross-talk and voltage transients on the power and ground planes. Signals to keep isolated are: clocks from everything else, digital and analog signals from each other, and high and low frequencies from each other.

Isolation can be achieved by spacing, running signals on separate board sections or layers, and isolating ground and power planes. With digital clock layout, minimum spacing and isolation with ground traces are often required to ensure a clean signal.

Analog signals are extremely susceptible to interface and corruption from noise. Analog and digital signals normally use different ground planes and often separate power planes. Isolation to different board sections and, preferably, different board layers is also considered good design practice. Layout of boards so that the analog ground is isolated from digital board sections helps minimize interference. When possible, run digital and analog signal traces on different board levels and isolate them by running a ground or power plane between them.

For proper layout of high- and low-speed digital signals, careful decoupling, and trace separation are required. Where possible, use separate board sections and different ground and power planes to minimize interference.

Cables and Cable Connectors

Cables and cable connectors have to be selected and routed with care to avoid problems. Improper cable and connector installation can cause EMI, cross-talk, and insertion loss. The connection can also suffer from mechanical stress and corrosion if not carefully implemented.

Cable

The quality of the cable has a significant effect on cross-talk and EMI. As with traces, cables have inductive, capacitive, and antenna effects. Selecting low-quality cables or bundling poorly shielded cables together can generate substantial cross-talk and EMI. Cabling designed for high frequency with a matched current return length and shielding will significantly decrease cross-talk and EMI. Two examples of closely matched return loop cables are twisted-pair and coaxial cables.

Cable lengths should be as short as possible without inducing mechanical stress, especially when the cables run outside of a system box. The combination of short cables and closely matched current return loop will minimize EMI radiation and susceptibility.

Care should be taken when bending cables for routing into a cage or chassis. Excessively sharp bends can stress the cables and lead to early failure. Cables should be clamped and the bends gradual where possible.

Cable Connectors

Cable connectors are necessary to allow the disconnection of a cable from a board or chassis. The use of high-quality connectors is critical. Every cable and connector combination introduces a potential impedance discontinuity, mechanical stress point, and corrosion point.

A low resistance, low impedance connection between the cable, the connector, and the connector sections is critical for signal integrity. High-quality connectors will use components and design that ensure that this low resistance and impedance path is maintained over time and repeated insertions.

An alternative to cable connectors are clamping shields. Clamping shields are used with cables that will be permanently attached to a board or chassis. The clamp creates a mechanical bond between the cable and board. The clamp also electrically connects the cable shield to the board ground plane. The cable signal conductors are soldiered to the board. Using a clamping shield removes a potential discontinuity introduced by a conventional connector. The disadvantage is that the connection is not easily removed.

When using any type of connector and cable combination it is important to use metals that are the same or close on the Galvanic Series Chart. Dissimilar metals in contact can generate a positive ion flow from one metal to the other, creating corrosion. If left untreated, corrosion can destroy the mechanical and electrical integrity of the connector and cable.

The Military Standard (MIL-STD) Galvanic Chart and Specifications is one source of many that list metal types and information about how various metals react with each other. Another source is Underwriters Laboratory (UL) specification UL-1950 electrochemical potentials between dissimilar metals.

Cables and Cable Connectors (Continued)

TABLE 1. Partial list of the Military Standard MIL-STD-889 Galvanic Table

Active (Anodic)

1) Magnesium	42) Stainless steel 410 (active)	54) Bronze 220
4) Zinc	33) Copper (plated, cast, or wrought)	55) Copper 110
7) Aluminum 2014-T3	34) Nickel (plated)	58) Molybdenum, Commercial pure
10) Cadmium (plated)	35) Chromium (Plated)	59) Copper-nickel 715
19) Aluminum 6061-T6	36) Tantalum	61) Stainless steel 202
27) Tin (plated)	44) Tungsten	70) Silicone Bronze 655
41) Stainless steel 430	49) Yellow Brass	88) Titanium 75A
29) Lead	51) Brass (plated)	90) Silver
30) Steel 1010	52) Nickel-silver (18% Ni)	91) Gold
31) Iron (cast)	53) Stainless steel 316L	92) Graphite

Noble (Less Active, Cathodic)

Summary

The physical design of a backplane results from the electrical and physical requirements. Application, speeds, and architecture will all help determine the minimum physical structure necessary. When the minimum physical requirements are known, the form factor can be selected. Application and performance requirements also help the designer decide if the backplane will be a full custom design, a completely packaged protocol, or something in between.

Once the performance requirements, protocol, and form factor are chosen, system layout can begin. Signal layout and choices for connectors and cables will play a large part in system reliability and signal integrity as well as in overall system performance.