Table of Contents

Section **page no.**

Index of Figures

Index of Tables

1.0 OBJECTIVE

This specification provides information and requirements for customer application of FCI products within the XCede® connector family. It is intended to provide general guidance for development of customer designs and application processes as they relate to this product family. It should be recognized that no single process will work for all customer applications and that customers should develop processes to meet individual needs. However, if the processes vary from the guidelines, FCI cannot guarantee acceptable results.

2.0 SCOPE

This specification applies to XCede® press-fit daughtercard receptacles and backplane headers

3.0 REFERENCE DOCUMENTS

- Applicable FCI product customer drawings
- FCI Product Specfication GS-12-588 (XCede® Connector System)
- FCI Repair Manuals (TBD)

4.0 DEFINITIONS

4.1 Fillets

An extension of the pad at the interface of the trace to the pad that will allow more pad area in the event that the pad to hole registration compromises the interconnect area. See Figure 15 for details. For further information regarding these routing guidelines, please contact FCI Applications Engineering.

4.2 Foils/Copper Weights

Copper foil is measured in ounces (or weight). Common copper weights are 0.5 ounces, 1.5 ounces, and 2 ounces (3 oz up to 10 oz are available by special order). 1oz = .0014”, 1.5oz = .0021”, 2oz = .0028”.

4.3 Pads/Lands/Annular Ring

A pad is the support around a hole. If you see a specification calling out an annular ring of .005”, this indicates the amount of the pad left around the hole after processing.

4.4 Spacing

The space between two electrical connections, which can be defined as two lines, two pads, a line and a pad, etc.

4.5 Trace/Circuit/Line-Width/Lines/Conductor

Various different terms for an electrical connection. If you see the term .008” lines (or 8 mil lines), it means the electrical connection from one point to another will measure .008” in width.

4.6 Backplane

When used within this document, “backplane” refers to a fixed PCB in a chassis. The male (vertical header) half of the mating connector system is typically mounted to the backplane.

4.7 Daughtercard

When used within this document, “daughtercard” refers to a removable plug-in card within a chassis. The female half of a mating connector system is typically mounted on the daughtercard.

4.8 Monoblock Assembly

An assembly consisting of various daughtercard receptacle modules combined in a metal organizer.

5.0 CONNECTOR DESCRIPTION

5.1 Daughtercard Receptacle

The XCede® daughtercard receptacle modules are end-to-end stackable and can be configured in various combinations of signal, power, and guide modules. Unlike the backplane header modules, which are provided individually, the daughtercard receptacle consists of individual modules joined together into a custom monoblock assembly by means of a stamped metal organizer. Within this monoblock assembly could be a range of modules, such as XCede® high speed differential signal, power, guide, spacer, and low-speed signal. Daughtercard connector configurations are determined by the customer's specific system application requirements.

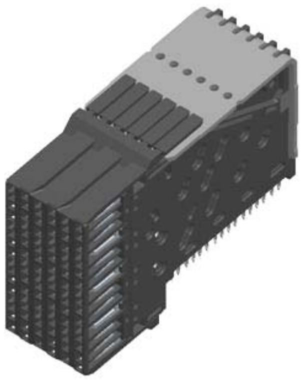


Figure 1: XCede® Individual Daughtercard Receptacle Module (Reference Only)

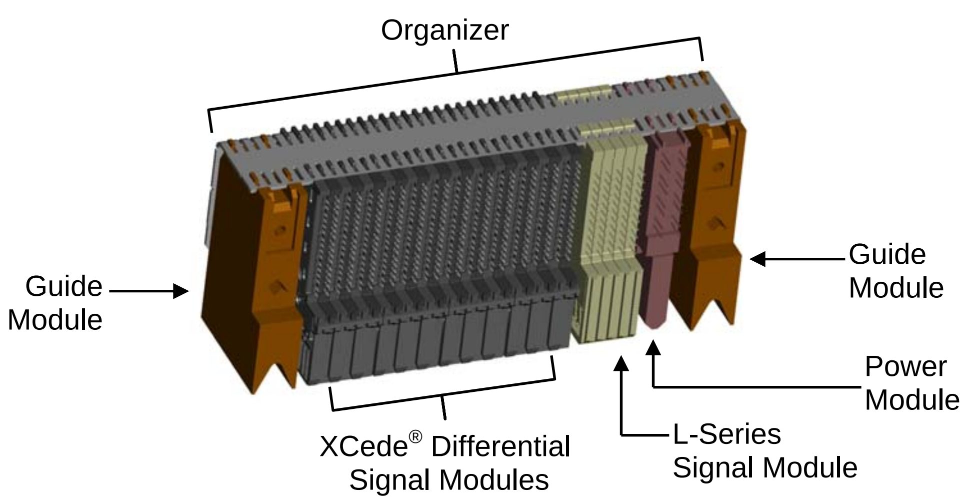


Figure 2: Hypothetical Daughtercard Receptacle Monoblock Configuration

5.2 Backplane Header

The XCede® backplane header modules can be configured in various combinations and are end-stackable. The modules are available in the following basic configurations:

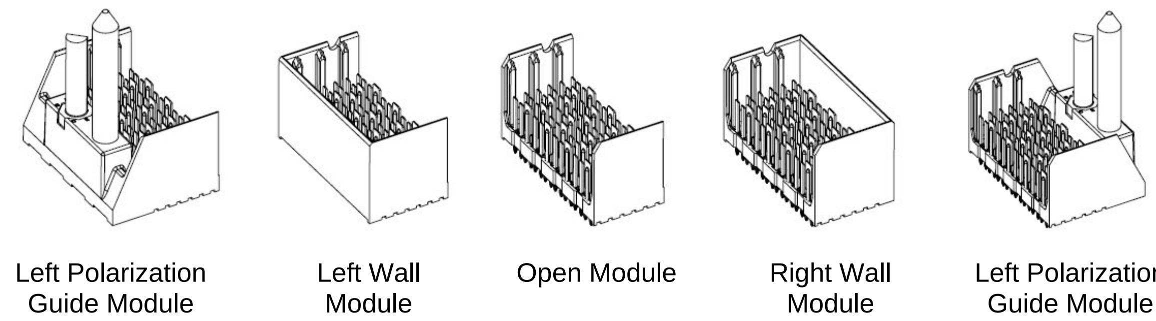


Figure 3: Available Header Signal Module Configurations

These modules can be combined with compatible power modules, low speed signal modules, and standalone guide pins as desired for a total stacked solution. Installation of keyed polarization pins in the end modules is optional.

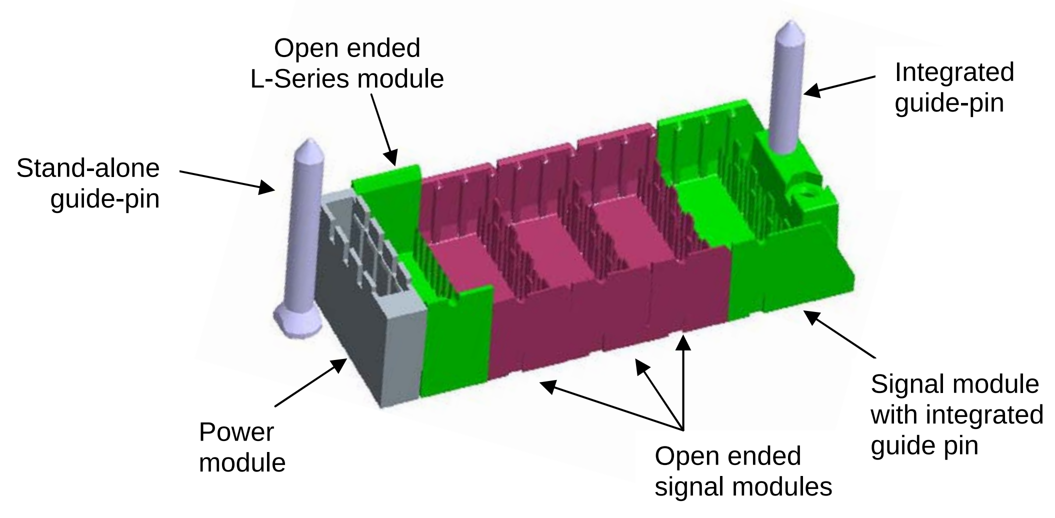


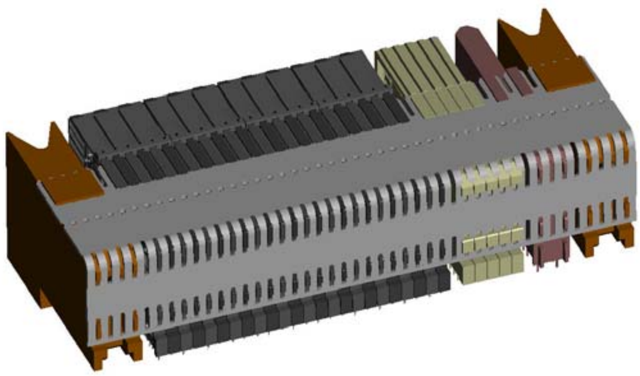
Figure 4: Hypothetical Backplane Header Configuration

6.0 CONFIGURATION GUIDELINES

6.1 Daughtercard Receptacle

6.1.1 General

Individual daughtercard receptacle modules are designed to fit into a stamped metal organizer. A variety of signal modules, guide modules, power modules, and other components can be assembled to the organizer on a 1.85mm pitch in the specific configuration required by the customer. The resulting connector is termed a "monoblock assembly". The configuration is typically determined by the design of the backplane slot.



6.1.2 Organizer

The metal organizer can help to straighten the board edge, but may not eliminate the need for additional board stiffening.

6.1.3 Optimal Daughtercard Connector Monoblock Configuration

It is recommended that the connector modules be configured in grouped increments of 6 wafers (or one signal module). Guide and power modules are typically placed at the end of the monoblock assembly to protect the daughtercard connector modules from handling damage.

6.1.4 Daughtercard Connector Monoblock Length Limitations

The minimum length of a daughtercard connector monoblock assembly is determined by the minimum number of signal modules and other components.

- ***The minimum number of signal modules for a standard configuration is 3.***
- ***The minimum number of power modules in a monoblock assembly is 3 when used as a standalone power assembly with no other assembled components.***
- ***The maximum length of a single daughtercard connector monoblock assembly (including the metal organizer) is 304.8mm (12 inches)***

This is constraint is based on the maximum length of the organizer component. The effective maximum usable length for the various assembled modules is 303.5mm.

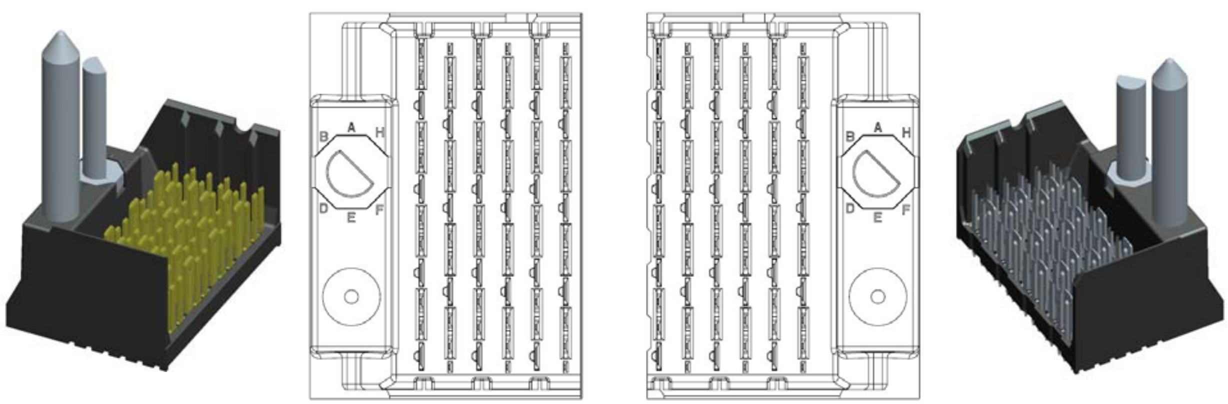
6.2 Backplane Header

6.2.1 General

Backplane header modules are specified and packaged individually. Signal modules, guide pins, power modules, and other components are customer-specified to match the configuration of the receptacle monoblock assembly of the mating daughtercard, then arranged and applied to the backplane accordingly. Integral guide-pins and keying can be included in the specification of backplane header end-modules.

6.2.2 Polarization (Keying)

A polarization (keying) pin is optional and can be configured in the end modules through a dash-number code the product part-number. The following figures show the keying pin and letter-designations for the 8 different orientations.



Left Polarizing Guidance Module

Right Polarizing Guidance Module

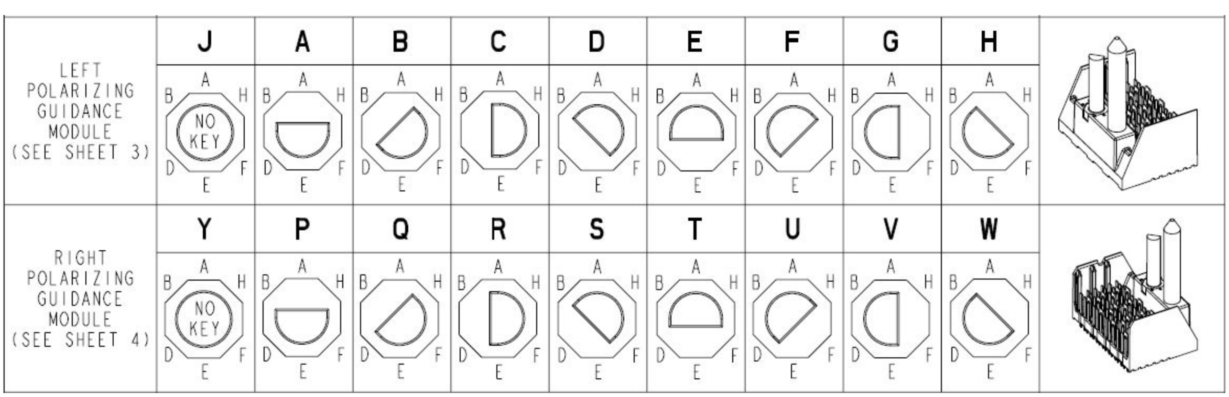


Figure 5: Polarization Orientations

6.3 Guidance

It is imperative that robust guidance be designed into any high-density high-pin-count connector system. The XCede® connector system relies on a sequential funneling system to ensure proper mating, the first phase being the gross alignment provided by the card-cage card-guides as the card approaches the backplane. The next level of alignment is provided by the XCede® guide-pins, with the final level of alignment contributed by the alignment ribs of the signal module front housing (receptacle) and associated alignment slots on the backplane header shroud. When considering guidance methods, many design considerations must be taken into account, for example:

- Length of connector
- Weight of daughter card
- Gathering ability of guide pins
- Orientation of connector in the application (horizontal or vertical)
- Amount of clearance in the card-guides
- The number of individual/different connectors on the same card edge

The length of the connector will help to determine the number of guide pins required. The following table provides further recommendations:

Daughtercard Connector Monoblock Assembly Length	Minimum Number of Guide-Pins Recommended
150mm or less	One guide-pin (minimum)
150mm – 360mm	Two guide-pins, one at each end of connector
360mm or greater	Contact FCI Applications Engineer

Table 1: Recommendations for Number of Guide-Pins

FCI does not recommend using more than three guide-pins due to the probability of “binding”, where the guide pins could work against each other due to tolerance stackup. Please contact FCI Applications Engineering for applications with unique requirements creating a need for more than there guide-pins.

There are three primary guidance systems available to compliment the daughtercard connectors:

1. Standard molded guide receptacle module (ungrounded or grounded for ESD protection)
2. Molded wide-guide receptacle module (ungrounded or grounded for ESD protection)
3. Die-cast wide-guide receptacle module

Likewise, there are three available guide-pin styles to compliment the backplane connectors:

1. Standard metal guide-pin, integral with applicable connector module
2. Standard stand-alone metal guide-pin, directly mounted to backplane
3. Large diameter metal guide-pin, directly mounted to backplane (for use with wide-guide receptacle)

The weight of the daughtercard assembly will also help to determine the best style of guide-pin or and guide-module used. See the following table and graph for the guide pin recommendations based on daughtercard weight, followed by a guide-pin force-deflection graph:

Weight of Daughtercard Assembly	Guidance System Recommendations
5 lbs or less	Use a minimum of one standard integral guide-pin and standard guide receptacle module.
5 lbs – 10 lbs	Option 1: Use two standard integral guide-pins and standard guide receptacle module. Option 2: Minimum of one standard stand-alone board-mounted guide-pin with standard guide receptacle module.
10 lbs – 15 lbs	Use two standard stand-alone board-mounted guide-pins with standard guide receptacle modules.
15 lbs or greater	Use large diameter stand-alone board-mounted guide pins with wide-guide receptacle modules.

Table 2: Guidance System Recommendations

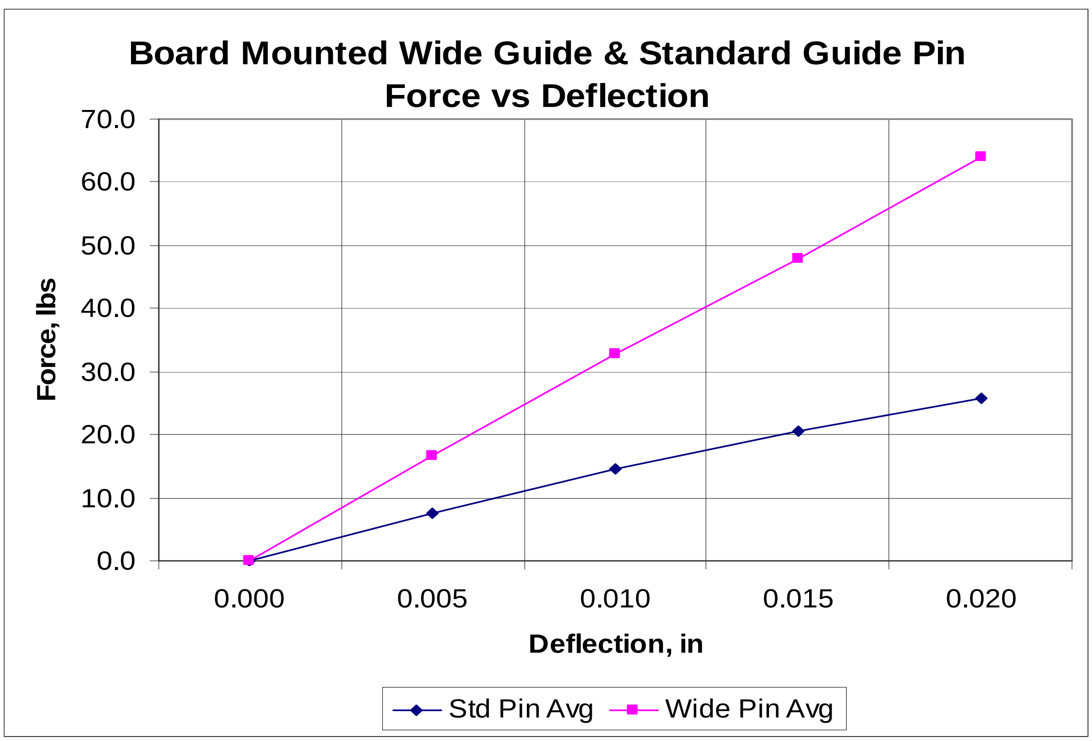


Table 3: Guide-Pin Force-vs-Deflection

The XCede® guidance system is capable of capturing a radial mating misalignment of 2mm with the standard guide-pins and 3mm with the large-diameter guide-pin as shown below:

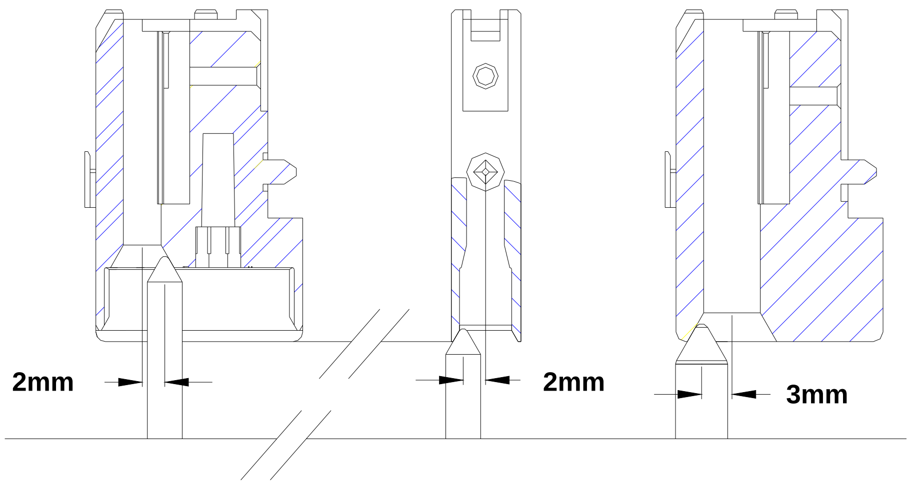


Figure 6: Gathering Capabilities of Guide-Pins

6.3.1 Signal Contact Lengths

There are two different contact length options available for each XCede® backplane header signal pair (please refer to applicable FCI customer drawing). Each option provides a specific mechanical mating wipe length. The minimum wipe length is calculated at the fully mated condition.

- The 4.20mm signal blade option provides a minimum of 2.0mm mating wipe length.
- The 5.20mm signal blade option provides a minimum of 3.0mm mating wipe length.

The standard configuration contains either “all long” or “all short” signal pair contacts within a module, however the backplane header can be configured with multiple signal pair blade lengths within the same module when a specific short-detect option is required.

6.3.2 Shield Contact Lengths

The backplane shield contact is available in one length, with a mating wipe of 4.0mm. The shield contact is intended to mate prior to all of the signal contacts, and to provide physical protection for all other contacts within the module.

6.3.3 Mating Sequence

Figure 7 shows relative mating sequencing between various components, along with the associated mating wipe distance.

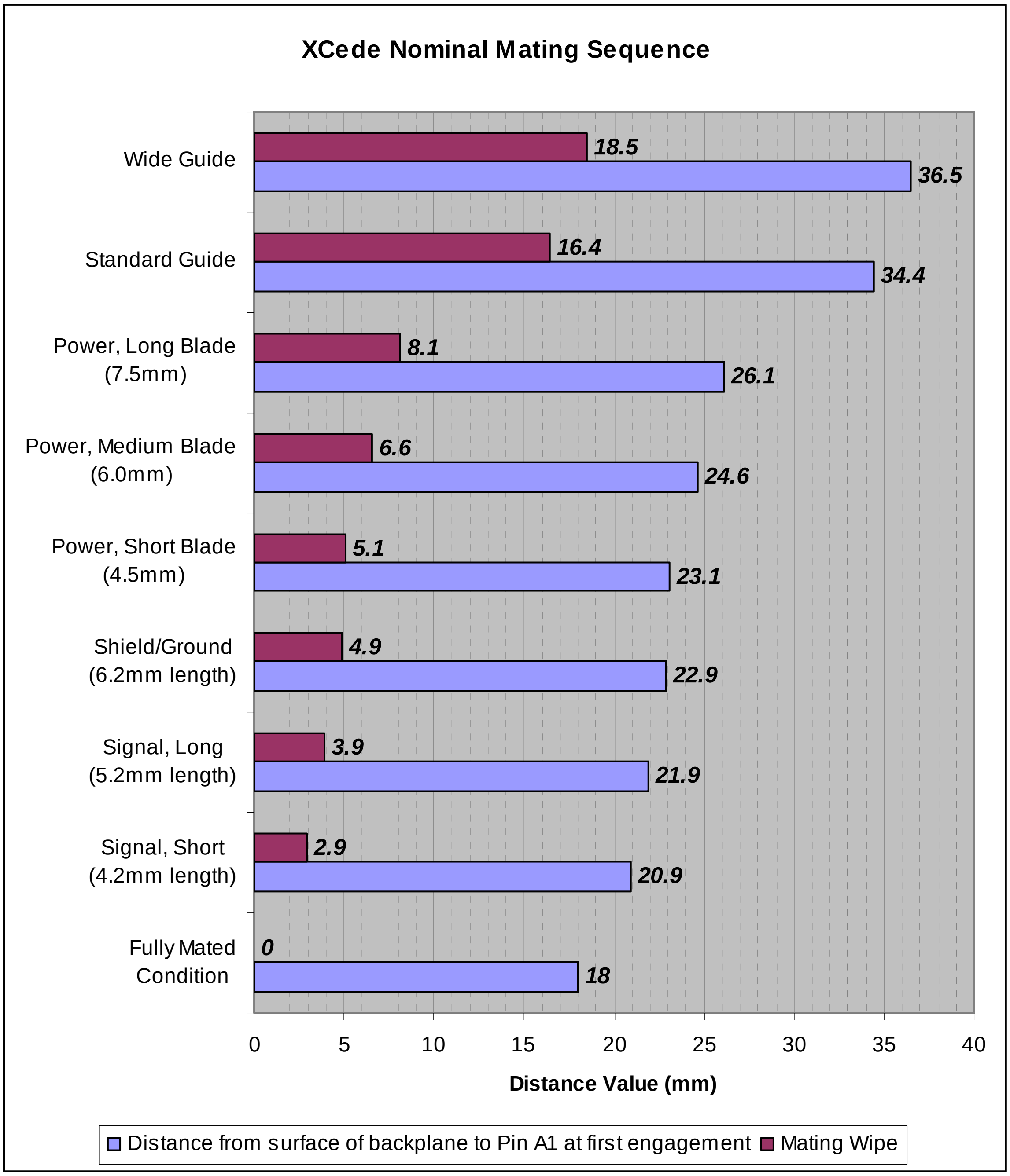


Figure 7: Mating Sequence and Wipe Distance

6.4 Compatibility With Hard-Metric Products

The recommended card-edge to backplane spacing for XCede® is 16.5mm (16.3/17.0mm depending on location of card-edge relative to pin A1 of daughtercard receptacle). The edge of the daughtercard must be notched when using this product in a hard-metric application, which requires a 12.5mm card-edge to backplane spacing. Representative FCI hard-metric products are ZipLine™, AirMax VS®, and Millipacs®.

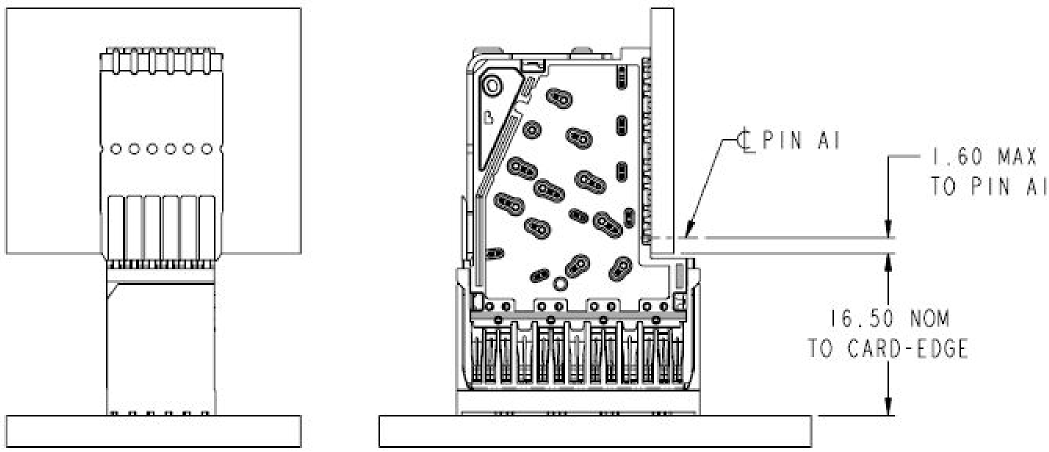


Figure 8: XCede® Card-edge to Backplane Distance

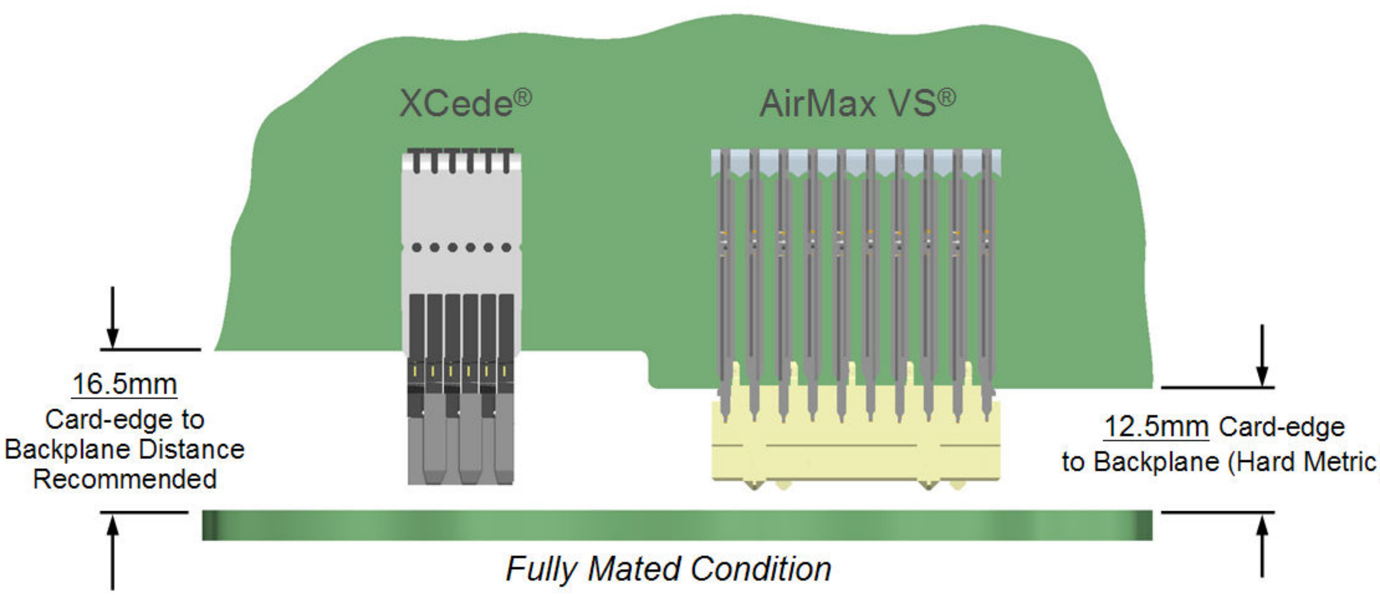


Figure 9: XCede® Card-Edge Requirements for Applications with Hard Metric Connectors

7.0 PCB DESIGN RULES AND MANUFACTURABILITY GUIDELINES

7.1 Footprint

For specific connector footprint, see customer drawings.

7.2 Minimum PCB Thickness

Long Tail (Daughtercard Receptacle): Recommended minimum PCB thickness of 0.063" (1.60mm)
Short Tail (Backplane Header): Recommended minimum PCB thickness of 0.035" (0.89mm)

7.3 Keep-Out Zones

Keep out zones are required around the connector footprint on the PCB for clearance with repair tooling that may be required. Please reference FCI customer drawings for dimensional specifications.

7.4 Hole Size and Finish

Reference Figure 12 & Table 6 for detailed dimensional recommendations regarding plated through holes.

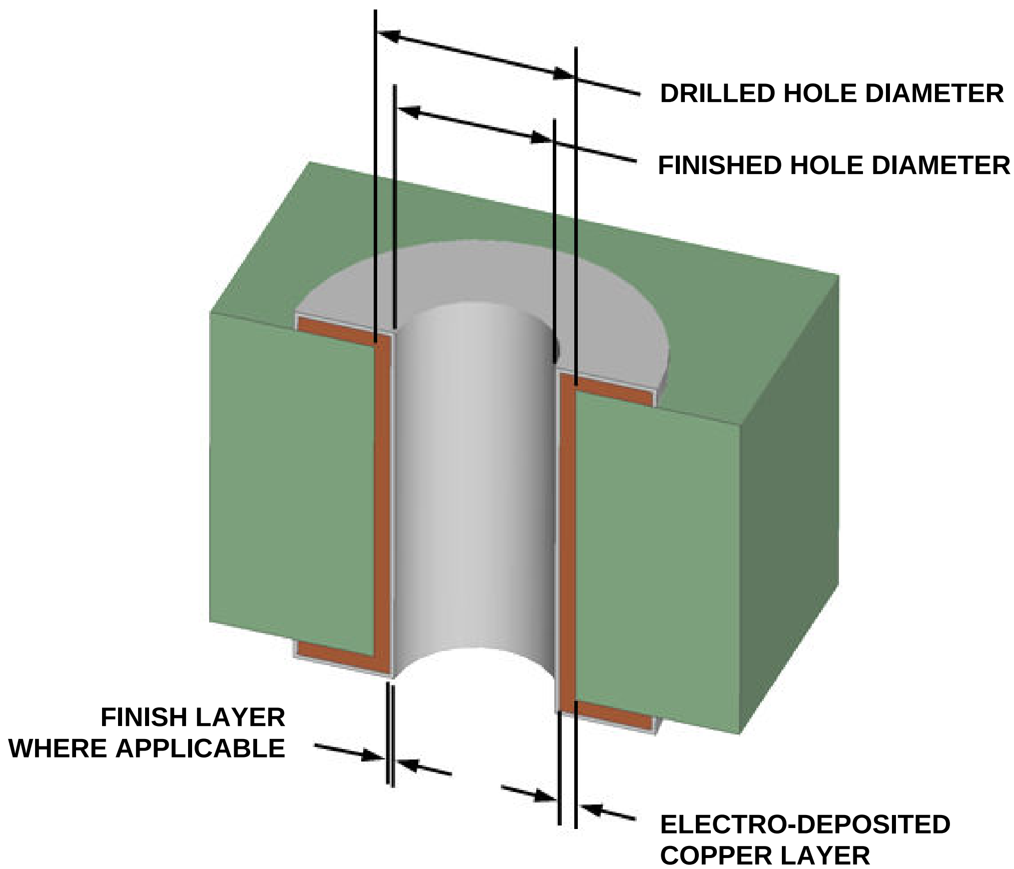


Figure 10: XCede® Typical Finished Hole Cross-Section

	PTH Finish Type				
	Solder Finish [(1)]	**Immersion Sn (Tin)**	**Immersion Ag (Silver)**	**Copper - OSP**	**Ni Au (Nickel – Gold)**
Drill Size	Long Tail (Daughtercard Receptacle): ISO 0.57mm (.0225") Drill #74 Short Tail (Backplane Header): ISO 0.55mm (.0217")				
Copper Thickness Per Side mm (in.)	0.0254 (.0010) min. 0.0635 (.0025) max.			0.0254 (.0010) min 0.0635 (.0025) max (DC) 0.0762 (.0030) max (BP)	0.0254 (.0010) min .0635 (.0025) max
Typical Finish Thickness	300-500µ"	35-70µ" min	4µ" min	N/A	53-210µ" Ni-Au compositions combined
Finished Hole Size	Long Tail (DC Recept): 0.46±0.05mm (.0180±.0020") Short Tail (BP Header): 0.45±0.05mm (.0177±.0020")				

1. *Solder finish includes: Tin/lead reflowed (plated and reflowed) and HASL*

Table 4: XCede® Plated Through-Hole (PTH) Requirements

7.5 Compliant Pin Critical Zone

There is a critical zone within the plated through hole where the dimensional requirements outlined in this specification and the applicable FCI customer drawings must be met in order for the compliant pin to function properly.

Conversely, in the area of the hole which is beyond this critical zone, the plated through hole may go below the minimum specified hole size *for non midplane applications*.

If backdrilling is used, it is allowed only in the area of the hole that is beyond the critical zone.

Please reference figures 12 & 13 for dimensional details regarding the compliant pin critical zone.

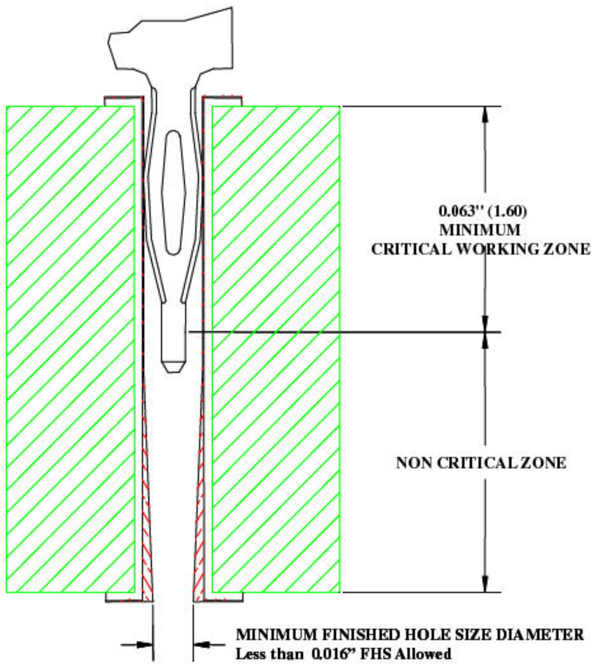


Figure 11: XCede® Long Tail Compliant Pin Critical Zone

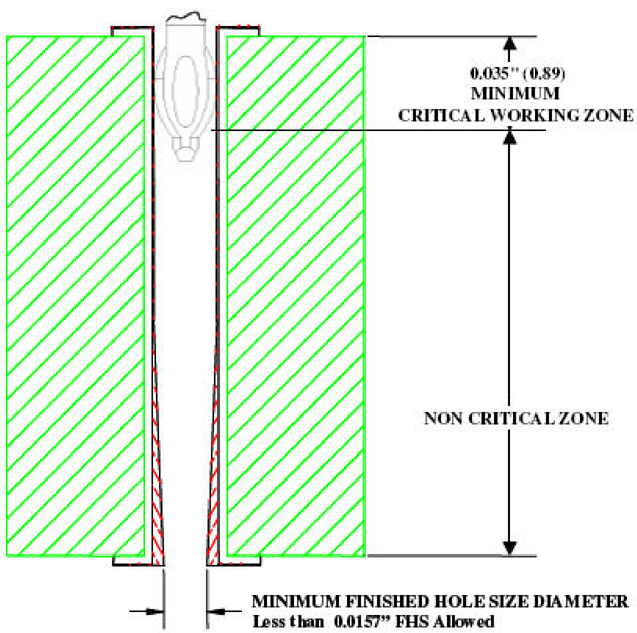


Figure 12: XCede® Short Tail Compliant Pin Critical Zone

7.6 Dual Diameter Via

The short tail compliant pin is particularly suited for use with dual-diameter vias, which should be considered for high-speed signals. The larger diameter is drilled first to a controlled depth followed by the second smaller drill.

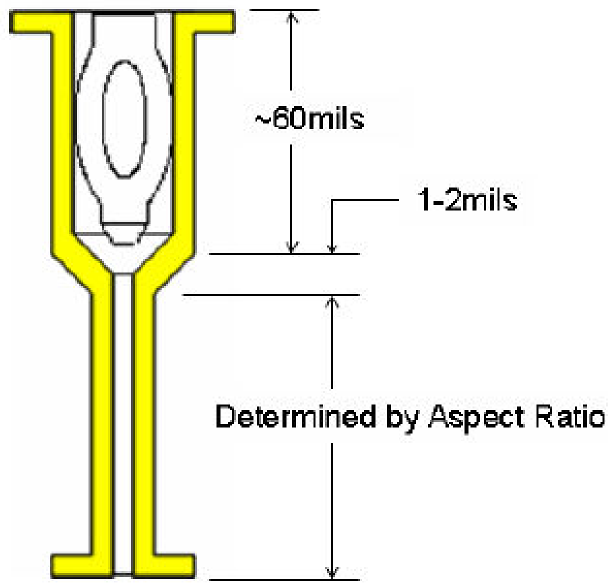


Figure 13: Diagram of a dual diameter via with XCede® short tail compliant pin

8.0 PCB ROUTING GUIDELINES

8.1 Surface Traces

Surface traces are not recommended. If surface traces are used, refer to applicable customer drawings for keep-out zones.

8.2 Line Widths, Pad Sizes and Spacing

Line widths, pad sizes and spacing are applicable for ½ ounce and 1 ounce copper weights.

8.3 Plane Clearances

Plane clearances are applicable for copper weights up to 2 ounces. Please contact FCI Applications Engineering for applications with more than 2 ounce copper.

8.4 Non-functional Pads

Non-functional pads on signal can be removed at designer's option.

8.5 Filleting

Filleting of pads is recommended (to be added by board fabrication house) for 0.000" annular ring (tangency), see Figure 11.

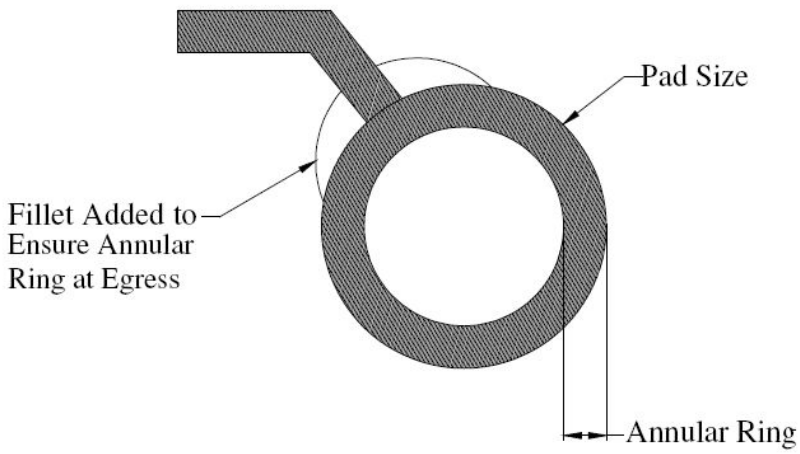


Figure 14: Preferred Fillet

8.6 XCede® Short Tail Signal Pad Sizes (see also Appendix A)

PCB Material Thickness (mm)	Copper Weight (ounces)	Process	Min. Pad Size 0.000 A/R in. / (mm)	Min. Pad Size 0.001 A/R in. / (mm)	Min. Pad Size 0.002 A/R in. / (mm)
0.062 to 0.400 (1.57 to 10.2)	0.5 (17µm)	Inner Layer	.034 (0.86)	.036 (0.91)	.038 (0.97)
		Outer Layer	.036 (0.91)	.038 (0.97)	.040 (1.02)
0.062 to 0.400 (1.57 to 10.2)	1.0 (35µm)	Inner Layer	.035 (0.89)	.037 (0.94)	.039 (1.00)
		Outer Layer	.037 (0.94)	.039 (0.99)	.041 (1.04)
0.062 to 0.400 (1.57 to 10.2)	2.0 (70µm)	Inner Layer	.037 (0.94)	.039 (0.99)	.041 (1.04)
		Outer Layer	.039 (0.99)	.041 (1.04)	.043 (1.09)

1. *Outer layer pad sizes reflect panel plating process.*
2. *User inner layer pad sizes for outers when pattern plating.*
3. *Values in ( ) are metric equivalents. For printed circuit board layout, use metric units.*
4. *For plane clearances, see Section X.X. For all other plane clearances, contact FCI Applications Engineering.*

Table 5: Short-Tail Signal Pad Sizes

8.7 XCede® Long Tail Signal Pad Sizes (see also Appendix B)

PCB Material Thickness (mm)	Copper Weight (ounces)	Process	Min. Pad Size 0.000 A/R in. / (mm)	Min. Pad Size 0.001 A/R in. / (mm)	Min. Pad Size 0.002 A/R in. / (mm)
0.062 to 0.400 (1.57 to 10.20)	0.5 (17µm)	Inner Layer	.035 (0.89)	.037 (0.94)	.039 (0.99)
		Outer Layer	.037 (0.94)	.039 (0.99)	.041 (1.04)
0.062 to 0.400 (1.57 to 10.20)	1.0 (35µm)	Inner Layer	.036 (0.91)	.038 (0.97)	.040 (1.02)
		Outer Layer	.038 (0.97)	.040 (1.02)	.042 (1.07)
0.062 to 0.400 (1.57 to 10.20)	2.0 (70µm)	Inner Layer	.038 (0.97)	.040 (1.02)	.042 (1.07)
		Outer Layer	.040 (1.02)	.042 (1.07)	.044 (1.12)

1. *Outer layer pad sizes reflect panel plating process.*
2. *User inner layer pad sizes for outers when pattern plating.*
3. *Values in ( ) are metric equivalents. For printed circuit board layout, use metric units.*
4. *For plane clearances, see Section X.X. For all other plane clearances, contact FCI Applications Engineering.*

Table 6: Long-Tail Signal Pad Sizes

8.8 Typical Anti-pad Geometry

The following describes the nominal antipad design for XCede®. Variations to this design based on stackup design and performance requirements are allowed.

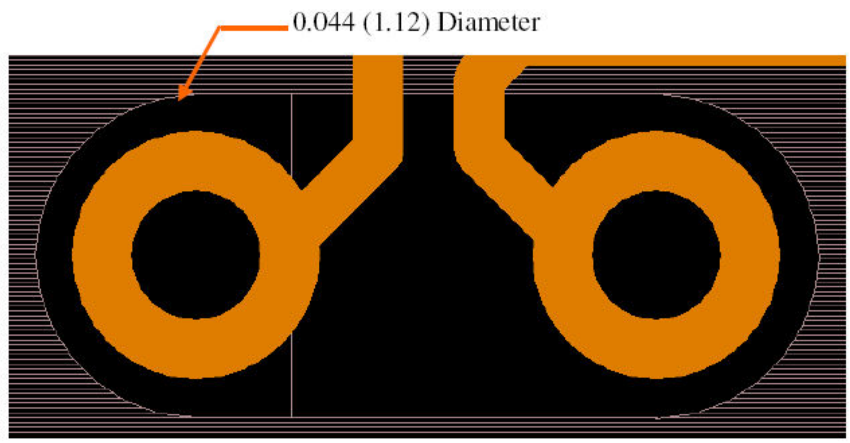


Figure 15: Antipad Clearance - XCede® Backplane Connector

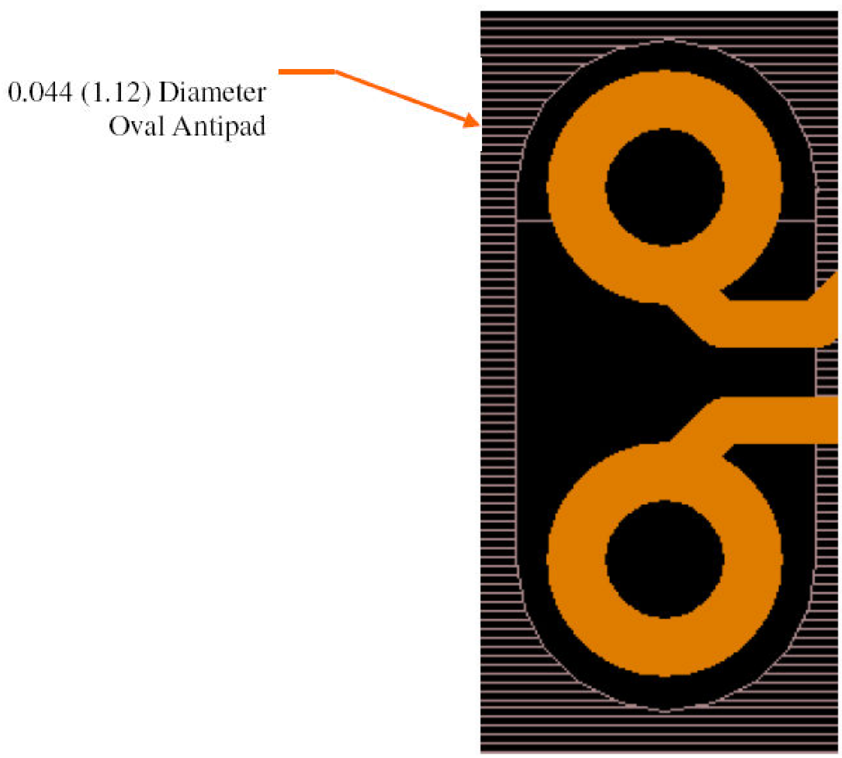


Figure 16: Antipad Clearance - XCede® Daughtercard Connector

8.9 XCede® High-Speed Differential Routing

For complete hole pattern dimensions, please refer to the FCI customer drawings.

8.9.1 Example: XCede® Daughtercard Routing

The available space for routing is determined by the distance between antipads. *There is no secondary routing channel available on the Daughtercard for differential pairs.*

Center-to-Center	.0728" (1.850mm)
- AntiPad Diameter	.044" (1.120mm)
= Resulting Space for Traces	**.0288" (0.731mm)**
Line Width	.007" (0.152mm)
Space Between Lines	.007" (0.178mm)
Resulting Ground Plane Web Overhang on Each Side	.004" (0.099mm)

Table 7: Calculation of Space Available for Daughtercard Routing

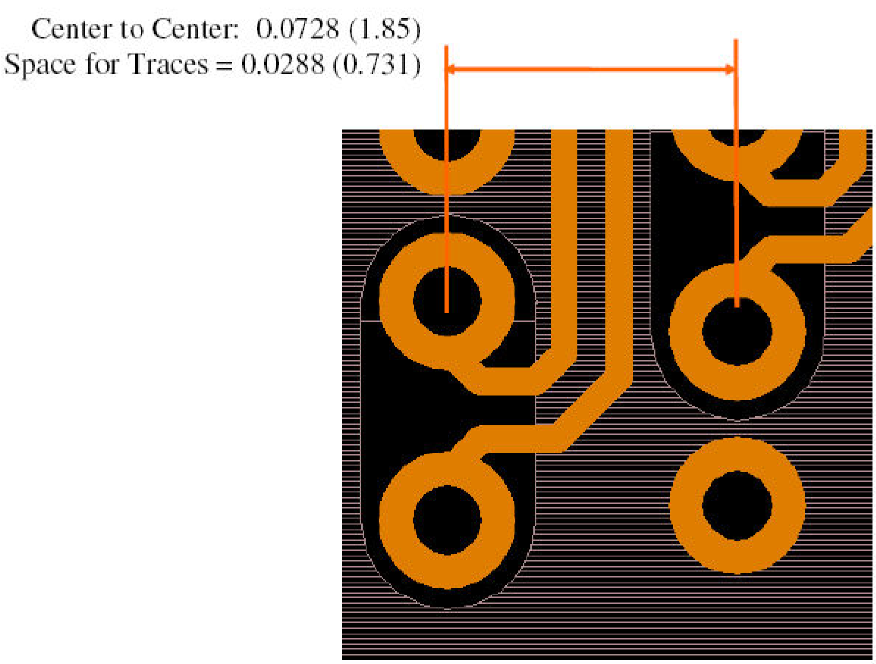


Figure 17: Example of XCede® Daughtercard High Speed Differential Routing

8.9.2 Example: XCede® Backplane Routing

The amount of space available for routing is determined by the ground plane web between the antipad clearances.

Center-to-Center	.0728" (1.850mm)
- AntiPad Diameter	.044" (1.120mm)
= Resulting Space for Traces	**.0288" (0.731mm)**
Line Width	.007" (0.152mm)
Space Between Lines	.007" (0.178mm)
Resulting Ground Plane Web Overhang on Each Side	.004" (0.178mm)

Ground Pin Center-to-Center	.06142" (1.560mm)
- Drill Diameter	.0217" (0.55mm)
- Required Clearance to Barrel X 2	.018" (0.46mm)
= Resulting Space for Traces	**.0217" (0.731mm)**
Line Width	.007" (0.152mm)
Space Between Lines	.007" (0.178mm)

Table 8: Calculation of Space Available for Backplane Routing

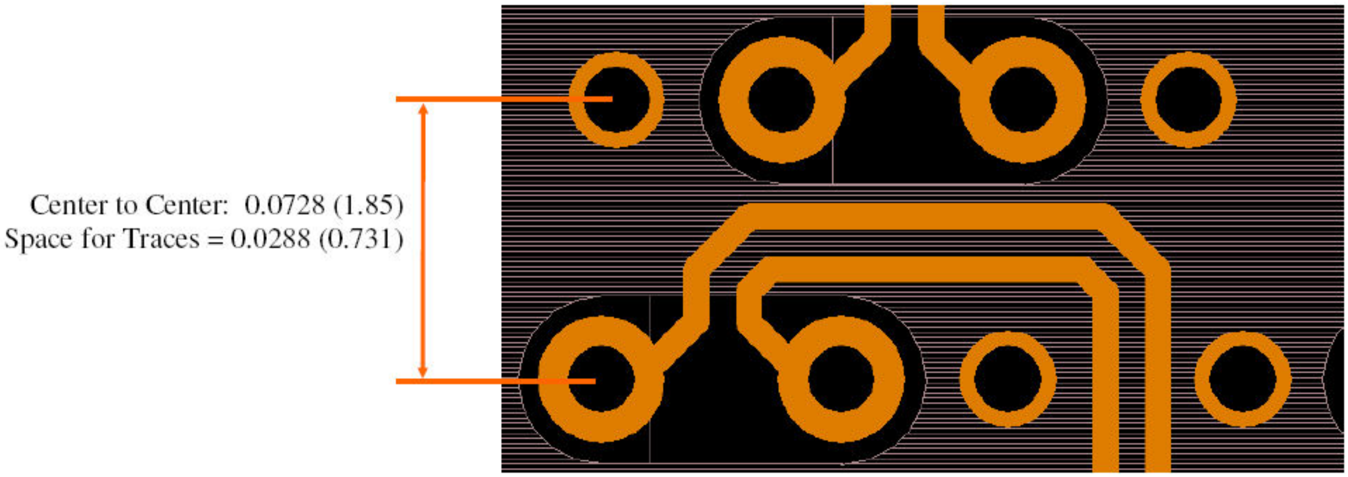


Figure 18: Example of XCede® Backplane Typical High Speed Differential Routing – Ground Plane

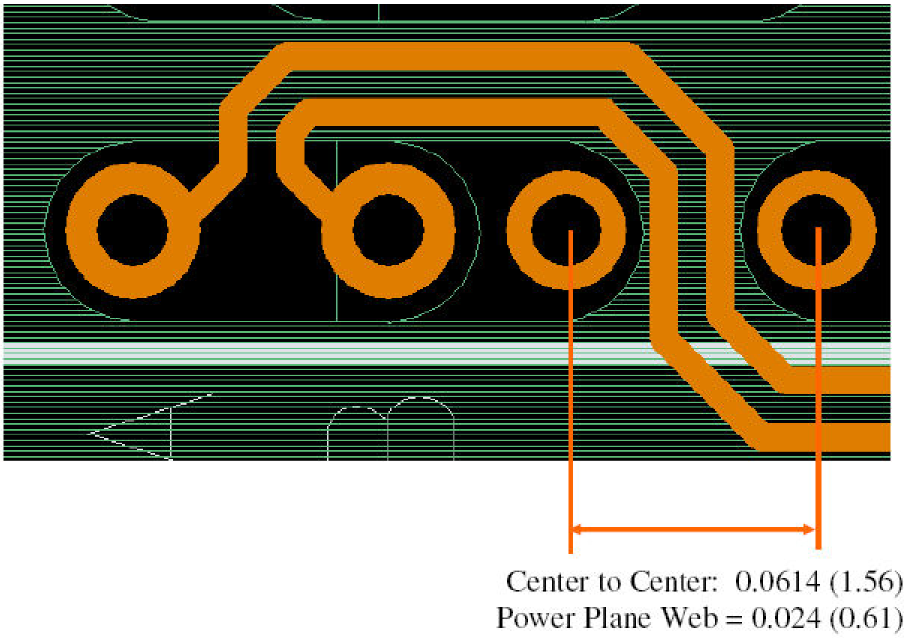


Figure 19: Example of XCede® Backplane Typical High Speed Differential Routing – Power-Plane

8.9.3 Elongated Antipad for Improved Return Loss

The double ground vias between signal pairs in XCede® allow for elongated antipads that directly pass through the ground vias. A rectangular shape example is shown below in Figure 22. Other shapes are allowable.

Figure 20: Example of Elongated Rectangular Backplane Antipad clearance

Figure 22 shows the performance improvement in S11 using the enlongated rectangular anitpad. This plot is from an HFFS simulation of a 24 layer backplane using Nelco 4000-13 with a 32mil stub.

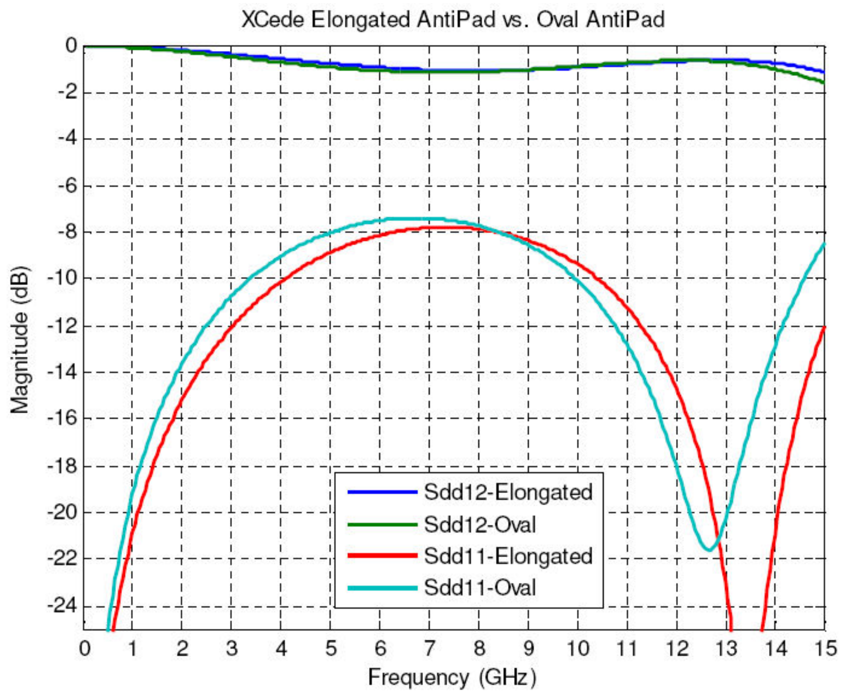


Figure 21: Peformance Improvement of Elongated Antipad

8.9.4 2-Layer Escape Routing with the XCede® 4-pair Connector

The following figure shows how to escape out of the 4-pair connector in only 2 layers. If this strategy is used, space for 11 pairs between connector modules is needed. Six open teeth in the stamped organizer are needed to fit 11 pairs using 6-7-6-24 spacing in N4K13 material.

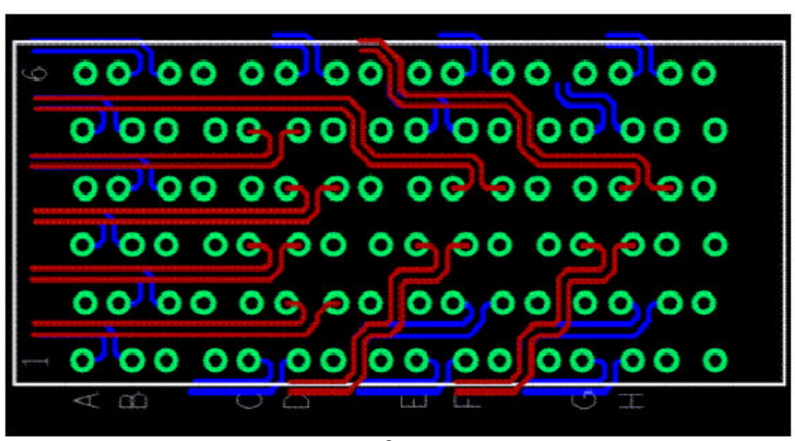


Figure 22: Example of XCede® 4-Pair 2-Layer Escape Routing

8.10 Midplane Applications

The XCede® connector can be used in a midplane application where vias are “shared” between front and rear cards. The front and rear cards must be “coplanar”, meaning that the front and rear card-guides must line up. The connectors must be offset by one column and the front card pin numbering sequence is opposite of the rear card pin numbering sequence.

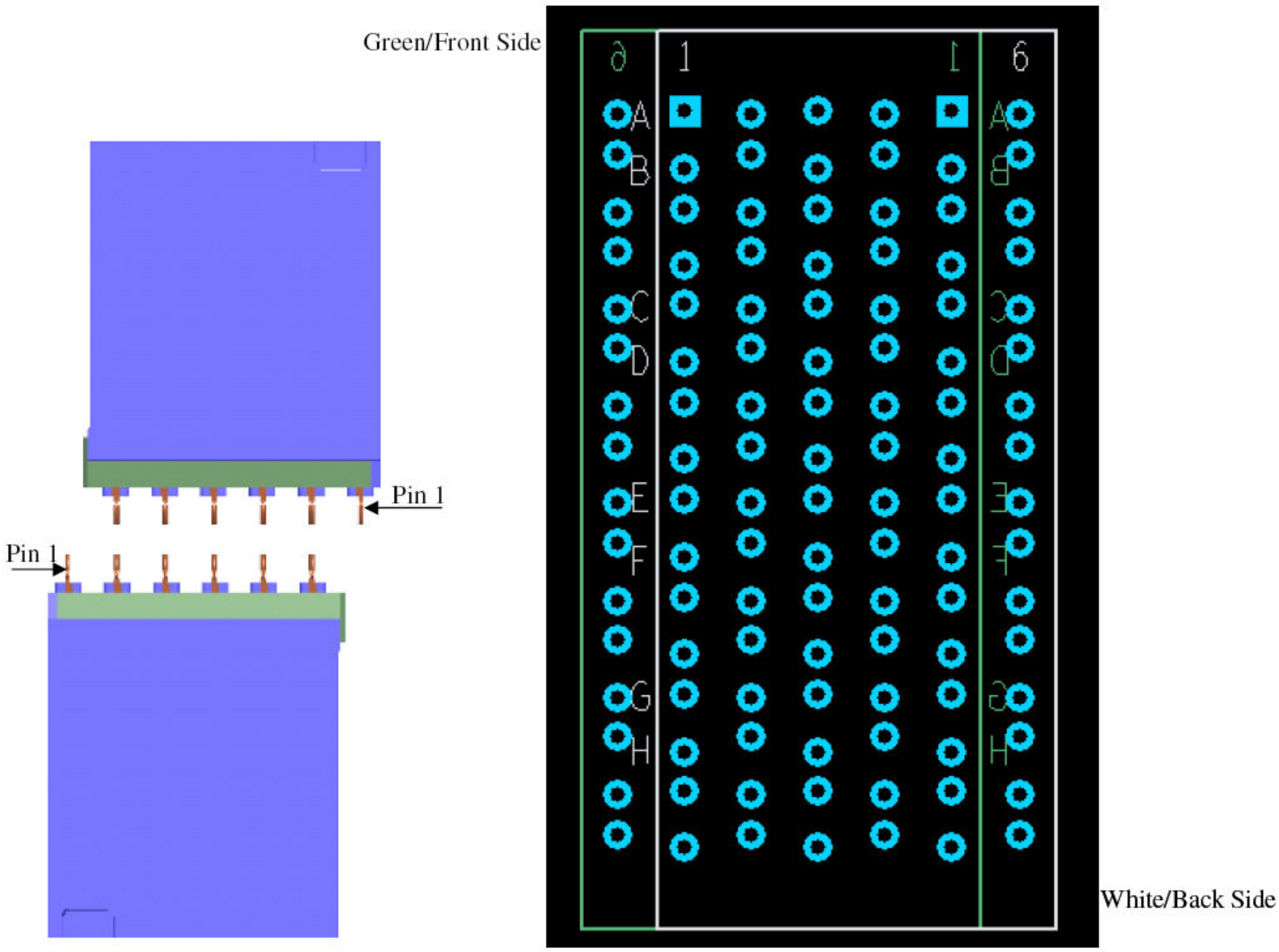


Figure 23: Example of XCede® Midplane Connector Application

9.0 INSTALLATION REQUIREMENTS

9.1 Application Press System Requirements

9.1.1 General

The monoblock nature of the daughtercard receptacle requires special attention when choosing an application press. FCI recommends the ASG MEP 12T electric press for installation of daughtercard monoblock assemblies. Air or hydraulic presses are typically less suited for the controlled press rate and controlled force required of the daughtercard installation.

9.1.2 Force

The application press must have the capability of installing connectors with a maximum force of 44.5N (10lbs) per press-fit tail. Please refer to FCI product specification for specific performance criteria regarding press-fit tails.

9.1.3 Rate

The recommended press-head rate of travel is 1.27 mm/sec (.050 in/sec) with the appropriate application force.

9.1.4 Structure

The press, fixture, and tooling combination must be adequately rigid to minimize deflection during the pressing cycle. Forces must be transmitted directly to the connector without inducing any side load or moment to the connector assembly. The press must have a minimum Z-axis stroke of 19mm (.750”). The press bed must be parallel to the pressing surface of the insertion tooling.

9.1.5 Feedback

The application press should have the capability to monitor, display, and record insertion force data throughout an individual press cycle, allowing continuous monitoring and data collection to evaluate any failure or machine malfunction. Speed or height controls should also allow a temporary press cycle stop at a repeatable position with reference to the board surface, or with reference to the insertion force. The press should have the capability to detect force variations as low as 2.25N (10 lbs) during the pressing cycle, and adjust the press cycle based on force feedback.

9.2 Application Tooling

9.2.1 Daughtercard Receptacle

The application tool recommended for XCede® Daughtercard Receptacles is a simple press-block as shown in figure 25. Please refer to the following table for sizes and part numbers of this tooling.

Product Type	**Press-Block Length** (for corresponding monoblock assembly length)	**Press-Block Part Number**
4-Pair	102mm / 4"	10102209-001
4-Pair	152mm / 6"	10102209-002
4-Pair	203mm / 8"	10102209-003
4-Pair	254mm / 10"	10102209-004
4-Pair	305mm / 12"	10102209-005

Table 9: Daughtercard Receptacle Press-Block Part Numbers

For any given application, the chosen press-block length should be equal to, or greater than, the total length of the monoblock daughtercard assembly.

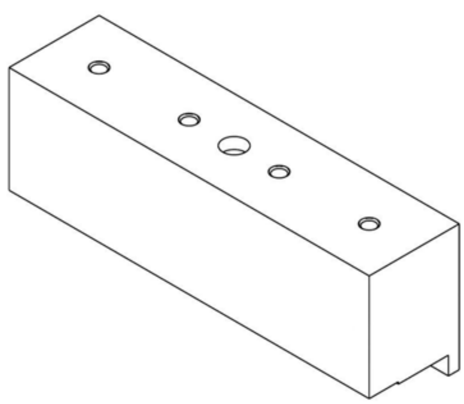


Figure 24: XCede® Daughtercard Receptacle Press-Block

Application of the daughtercard receptacle monoblock assembly requires an appropriate backup fixture to support the PCB during the connector installation process. This fixture is to provide adequate support directly under the connector assembly based on the footprint pattern. The customer-designed support fixture should accommodate any compliant pin lead protrusion on the secondary side of the PCB and should minimize any unsupported span. The selected material should allow minimal deflection under the maximum compression force. Please contact your FCI applications engineer for any needed assistance.

9.2.2 Backplane Header

The application tool recommended for XCede® Backplane Headers is a custom press-tool as shown in figure 27. Please refer to the following table for sizes and part numbers of this tooling.

Press-Tool Description	Press-Tool Part Number
Insertion Tool for 4-Pair, Vertical Header, 4-Columns	10102204-001
Insertion Tool for 4-Pair, Vertical Header, 6-Columns	10102204-002
Insertion Tool for 4-Pair, Vertical Header, 8-Columns	10102204-003
Insertion Tool for 4-Pair Vertical Header, 12-Columns	10102204-004

Table 10: XCede® Backplane Header Press-Tool Part Numbers

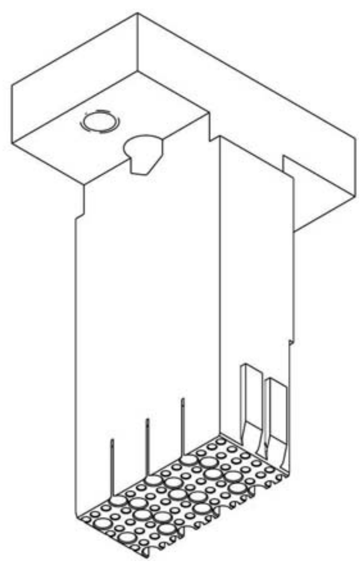


Figure 25: XCede® Backplane Header Press-Tool

(4-pair 6-column version shown)

Application of the backplane vertical header requires an appropriate backup fixture to support the PCB during the connector installation process. This fixture is to provide adequate support directly under the connector assembly based on the footprint pattern. The customer-designed support fixture should accommodate any compliant pin lead protrusion on the secondary side of the PCB and should minimize any unsupported span. The selected material should allow minimal deflection under the maximum compression force. Please contact your FCI applications engineer for any needed assistance.

9.3 Application Procedure: Daughtercard Receptacle

9.3.1 Connector Placement

Hold the connector just above the PCB hole pattern and angle the rear of the connector downward, placing the rearmost row of compliant tail tips into the correct plated through-holes (PTH). While maintaining engagement between the rear tails and the PCB holes, roll the front of the connector downward so that each successive row of compliant tails engages with the proper row of PTHs.

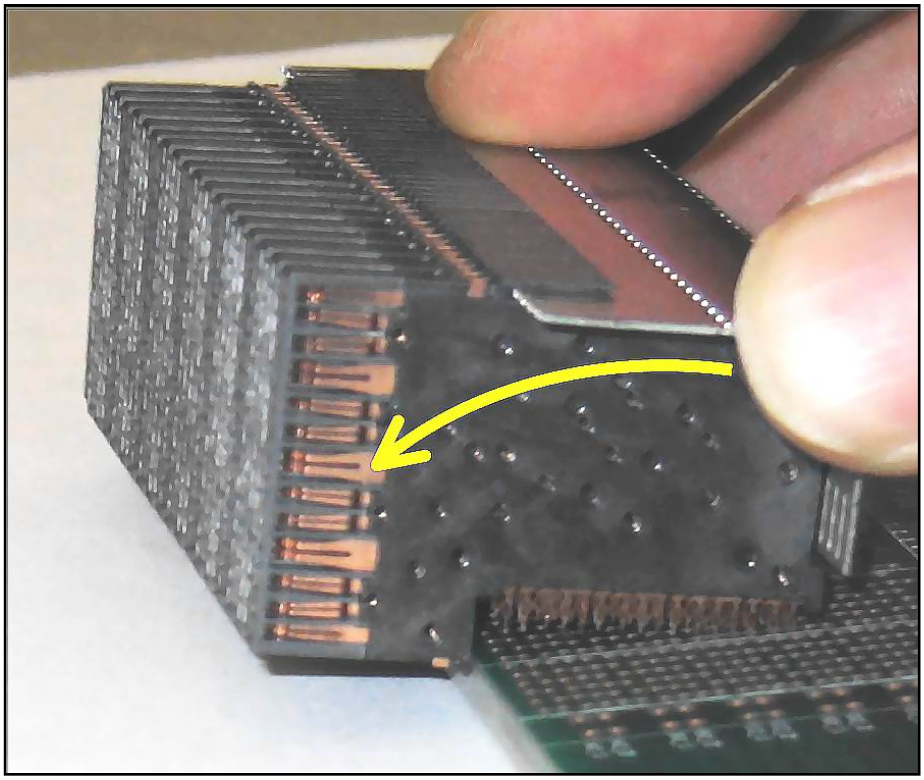


Figure 26: Manual Connector Placement - Daughtercard Receptacle

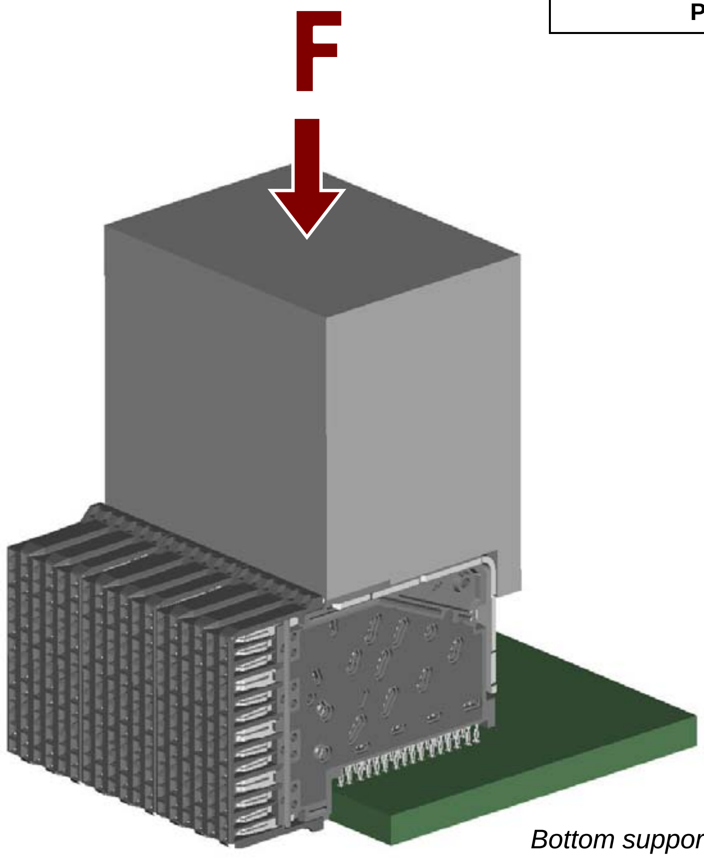


Figure 27: Press-Block Orientation - Daughtercard Receptacle

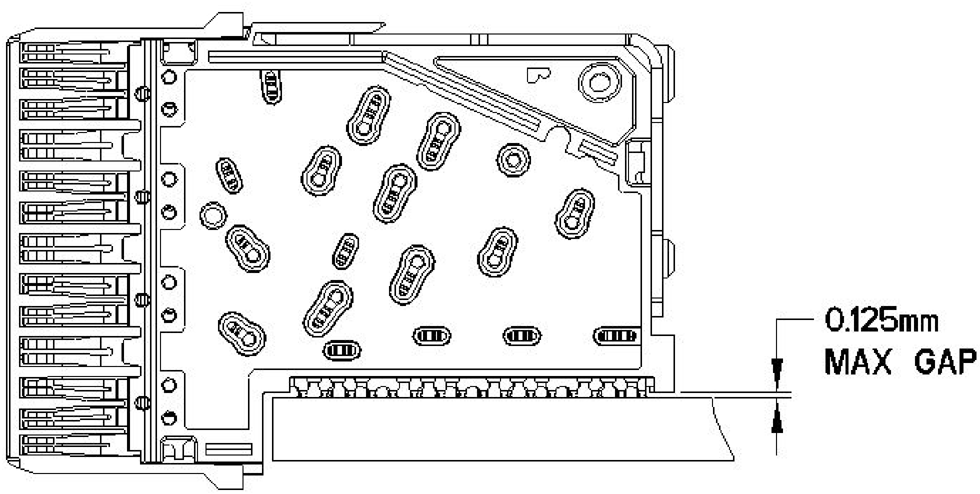


Figure 28: Fully Seated Inspection Dimension: Daughtercard Receptacle

9.4 Application Procedure: Backplane Header

Align the backplane header module tails with the applicable PCB holes and gently press by hand to stabilize and level the connector module. Carefully orient and align the applicable press-tool so that it freely clears all contacts and plastic alignment ribs. Press connector with adequate force based on insertion force requirements as detailed in the product specification.

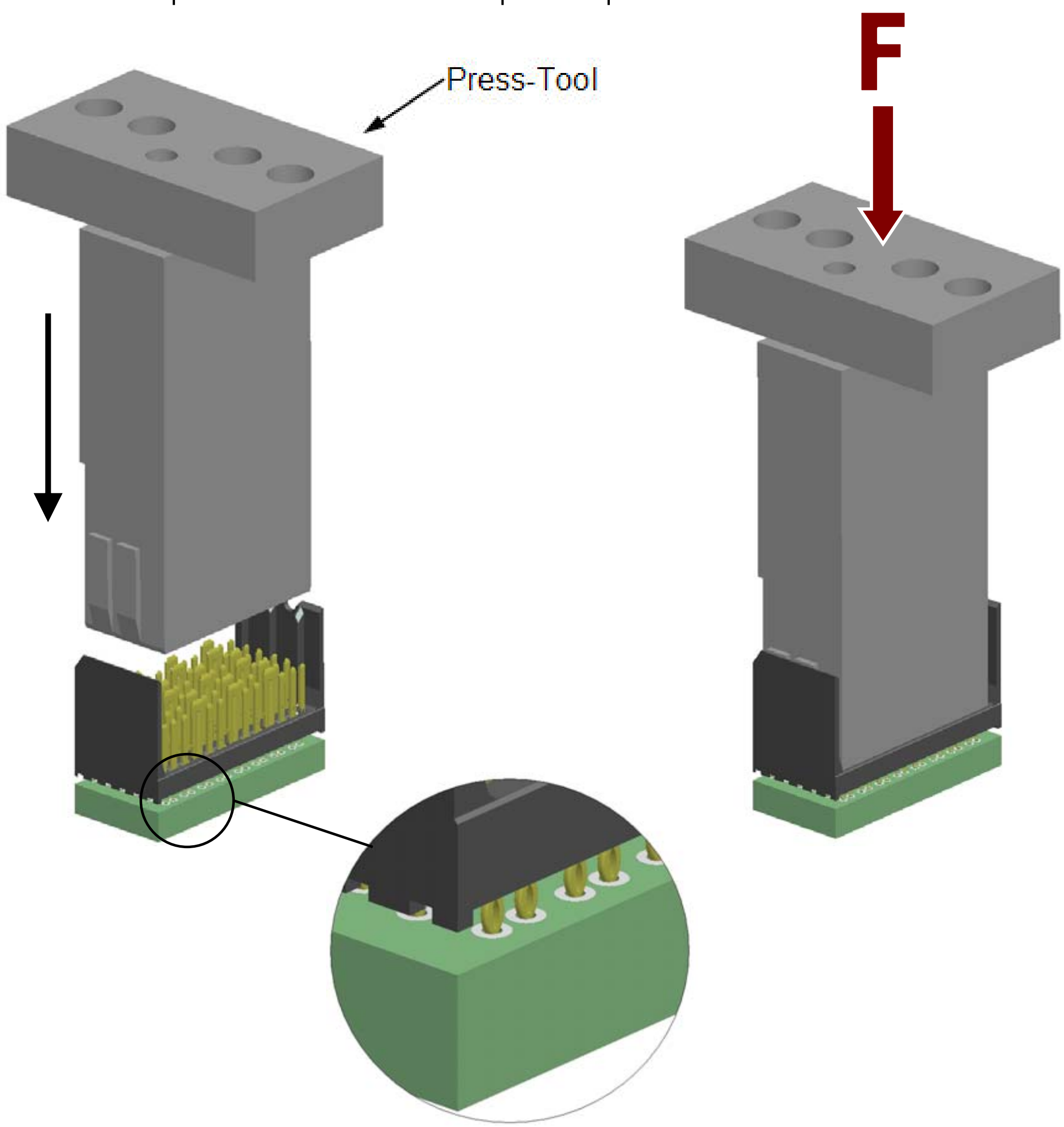


Figure 29: Module Placement and Press-Tool Alignment – Backplane Header

When the Backplane Header is fully seated on the PCB, any gap beneath the connector should be consistent on all sides, with a preferred gap of zero, but no greater than 0.20mm as shown in figure 30. This gap can be inspected with go/no-go shims of the appropriate sizes.

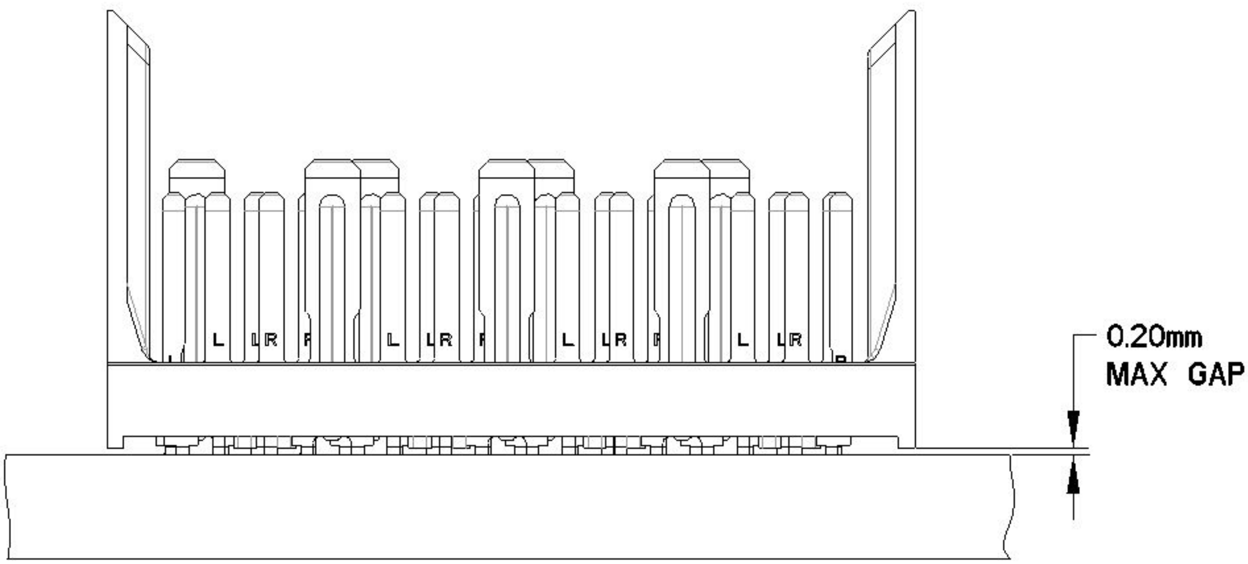


Figure 30: Fully-Seated Inspection Dimension – Backplane Header

10.0 REVISION RECORD

REV	PAGE	DESCRIPTION	EC #	DATE
1	all	Initial draft	-	2009-05-29