Table of Contents

Section **page no.**

1.0 OBJECTIVE

This specification provides information and requirements for customer application of FCI products within the Crossbow™ 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 Crossbow™ press-fit daughtercard receptacles and backplane headers to printed circuit boards (PCB's).

3.0 REFERENCE DOCUMENTS

- Applicable FCI product customer drawings
- FCI Product Specification GS-12-599 (Crossbow™ 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 10 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 oz, 1 oz, 1.5 oz, and 2 oz (3 oz up to 10 oz are available by special order). 1 oz = .0014", 1.5 oz = .0021", and 2 oz = .0028".

4.3 Pads/Lands/Annular Ring

A pad is the support around a hole. If you see a sp**e**cification calling out an annular ring of .005", this indicates the amount of the pad left around the hole after processing.

4.4 Spacing

Spacing is 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

These are 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/Midplane

When used within this document, "backplane" or "midplane" 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.

4.9 Wafer
An assembly module consisting of the metal contact structure with an over-molded plastic frame. These wafer modules are inserted into a daughtercard receptacle housing and attach to the metal organizer.

5.0 CONNECTOR DESCRIPTION

5.1 Daughtercard Receptacle
The Crossbow™ daughtercard receptacle modules 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 Crossbow™ orthogonal high speed differential signal, power, and guide. Daughtercard connector configurations are determined by the customer's specific system application requirements.

Figure 1: Crossbow™ 4x4 and 6x6 Daughtercard Receptacle Modules (Reference Only)

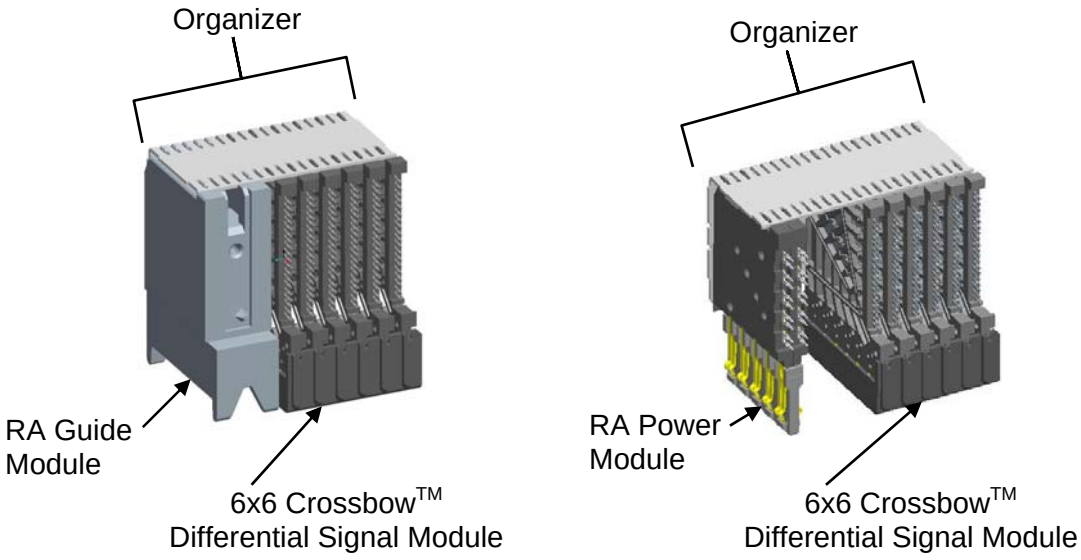
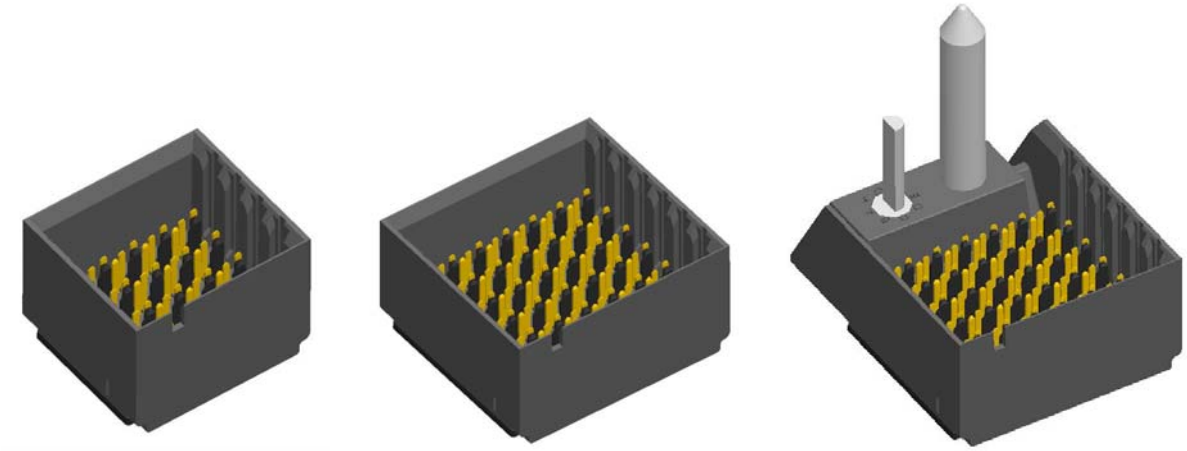


Figure 2: Hypothetical 6x6 Daughtercard Receptacle Monoblock Configurations

5.2 Backplane Header

The Crossbow™ backplane header modules can be configured in various combinations. Both 4x4 and 6x6 backplane headers have end walls. The modules are available in the following basic configurations:



4x4 4 wall Guide Module | 6x6 4 wall Guide Module | 6x6 3 wall Module With Guide

Figure 3: Available Backplane Header Signal Module Configurations

These modules can be combined with compatible power modules as desired for a total stacked solution. Installation of keyed polarization pins in the modules with guide pins is optional.

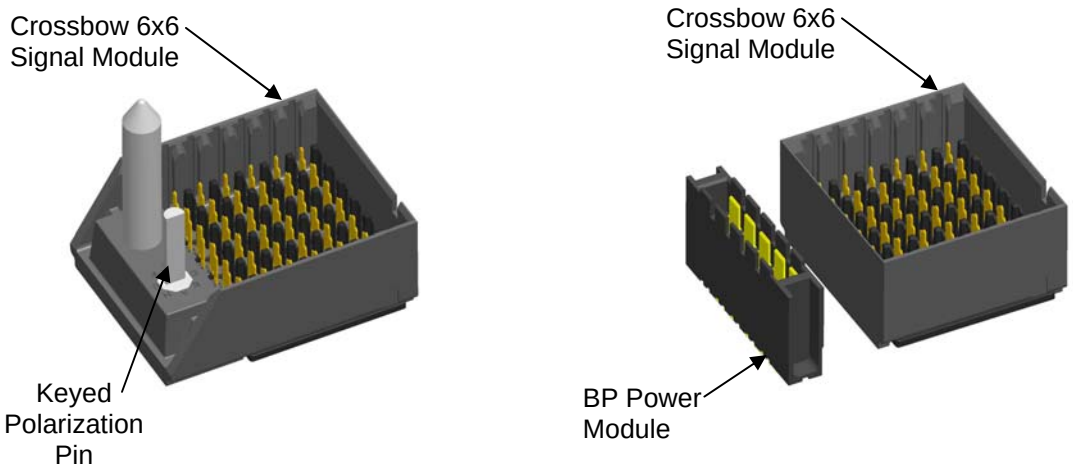


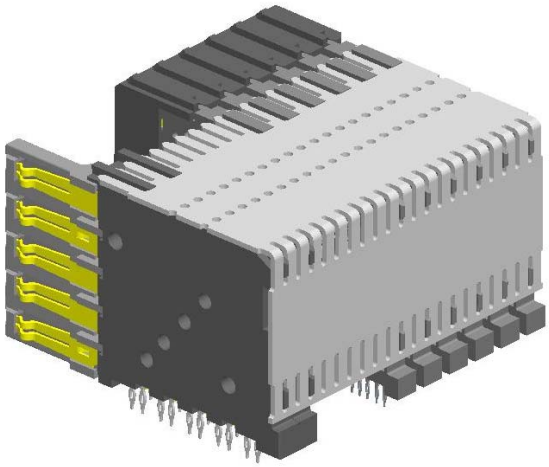
Figure 4: Hypothetical Backplane Header Configurations

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 2.00mm 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 Module Proximity Restrictions

4x4 and 6x6 assemblies cannot be mixed on the same organizer. When used in an orthogonal architecture, the Crossbow connectors are typically used in blocks. It is important to note that the card pitch on side 1 of the midplane is equal to the connector pitch on side 2 of the midplane when using shared vias.

6.1.4 Module Spacing

Spacing is defined, as the distance from one wafer feature to that same feature on an adjacent wafer, and must be in multiples of 2.00mm. All component dimensions from the print are nominal, unless otherwise specified.

6.1.5 Contact Plating Thickness

Mixing plating thickness within an assembly is not allowed.

6.1.6 Isolated DC Polarization

Guide modules not adjacent to a standard Crossbow™ signal wafer grouping, must mate with a free-standing guide pin on the backplane (BP).

6.1.7 Number of Guide Pins

No more than (2) guide pins per slot are recommended. For more that (2) guide pins within a configuration, contact an FCI Engineer.

6.1.8 Custom Markings

Custom marking are not allowed on the daughtercard receptacle connector assemblies.

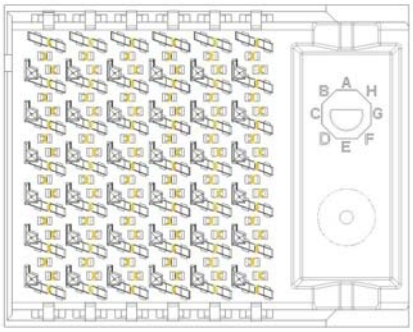
6.2 Backplane Header

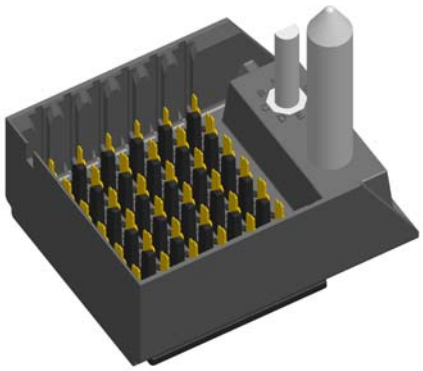
6.2.1 General

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 module with guide.

6.2.2 Polarization (Keying)

A polarization (keying) pin is optional and can be configured in the guide 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. A guide pin must be used with a polarization pin.





Right Polarizing Guidance Module

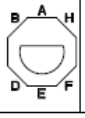
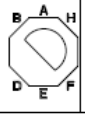
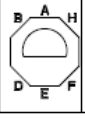
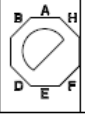
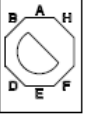
TABLE I									
PART NUMBER 10089485-XXXX	-X0XX	-XAXX	-XBXX	-XCXX	-XDXX	-XEXX	-XFXX	-XGXX	-XHXX
POLARIZING PIN ORIENTATION									

Table 1: Polarization Orientations

6.2.3 Signal Contact Lengths

There are two different contact length options available for each Crossbow™ 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 4.51mm signal blade option provides a minimum of 2.0mm mating wipe length.
The 5.51mm 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, (in pairs), within the same module when a specific short-detect option is required.

6.2.4 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.

6.2.5 Selective Contact Loading

All selectively loaded modules must be loaded in pairs. Different pin lengths within a pair are not allowed.

6.2.6 Contact Plating

Backplane contacts are offered with 30 and 50 micro inches of performance based plating. Within a module, different plating types cannot be mixed. A module must have only one type of plating on all pins.

6.2.7 Pin Depopulation

Depopulating signal pins, (in pairs only), is allowed in backplane header modules. Depopulating ground pins in backplane modules is also allowed.

6.2.8 Shared Via Holes

When sharing vias in orthogonal architecture, pin out sequences are called out on the FCI specification, (TBD). Backside connectors may share via holes using a 90 deg rotation or a 270 deg rotation. It is acceptable to use a 4x4 backplane module in a portion of shared via holes of a 6x6 footprint.

6.2.9 Common Ground Pins

In the backplane header module, the ground pins are common.

6.2.10 Custom Markings

Custom markings are not allowed on the backplane header connector assemblies.

6.3 Backplane Power Module

6.3.1 General

Backplane power modules are specified and packaged individually. Power modules are customer-specified to match the configuration of the receptacle monoblock assembly of the mating daughtercard, then arranged and applied to the backplane accordingly. A ten blade configuration is available in three lengths for sequential mating. Blade lengths must be selected in corresponding pairs.

6.3.2 Power Module Limitations

Crossbow utilizes the power module from another product platform. This other platform is based on 1.85mm organizer pitch whereas Crossbow has a 2.00mm organizer pitch. Therefore, only a signal power module can be used. Multiple power modules can only be used if extra slots are depopulated. No power modules are allowed back-to-back on midplane applications.

6.3.3 Power Contact Plating Thickness

There is only one gold plating thickness for backplane power modules, (50 micro inches).

6.3.4 Power Contact Sequencing

There are four levels of power sequencing, refer to figure 7.

6.4 Guidance

It is imperative that robust guidance be designed into any high-density high-pin-count connector system. The Crossbow™ 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 Crossbow™ 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

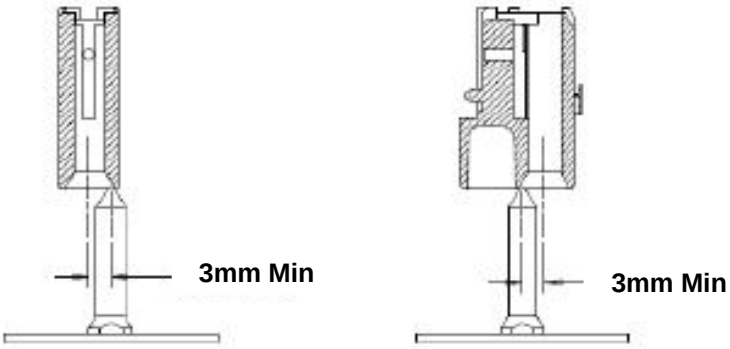
There are four primary guidance systems available to compliment the backplane connectors:

1. Plastic mount wide guide pin – 15 lbs max
2. Plastic mount wide guide pin with thru-hole screw – 30 lbs max
3. Board mount standard guide pin – 12 lbs max
4. Board mount wide guide pin – 25 lbs max

6.4.1 Guide Limitations

The backplane wide guide pin must mate with the corresponding daughtercard wide guide module. The wide guide has no polarization feature.

The Crossbow™ guidance system is capable of capturing a radial mating misalignment of 3mm with the large-diameter guide-pin as shown below:



Wide Guide /
Large-Diameter Guide-Pin

Figure 5: Gathering Capabilities of Guide-Pins

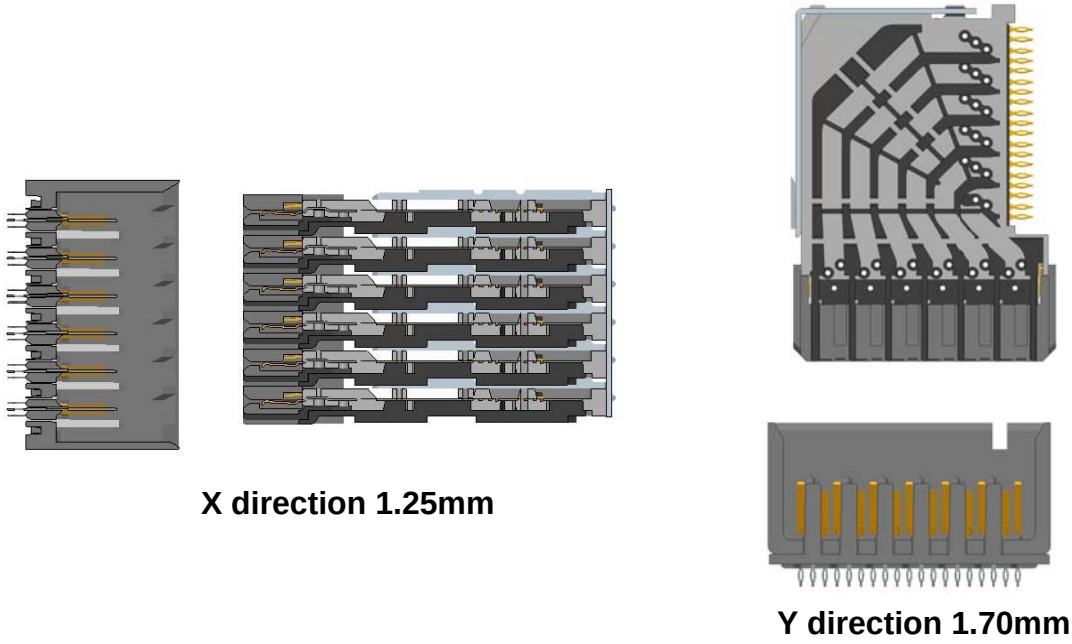


Figure 6: Gathering Capabilities of Crossbow™ Connectors

6.4.2 Mating Sequence

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

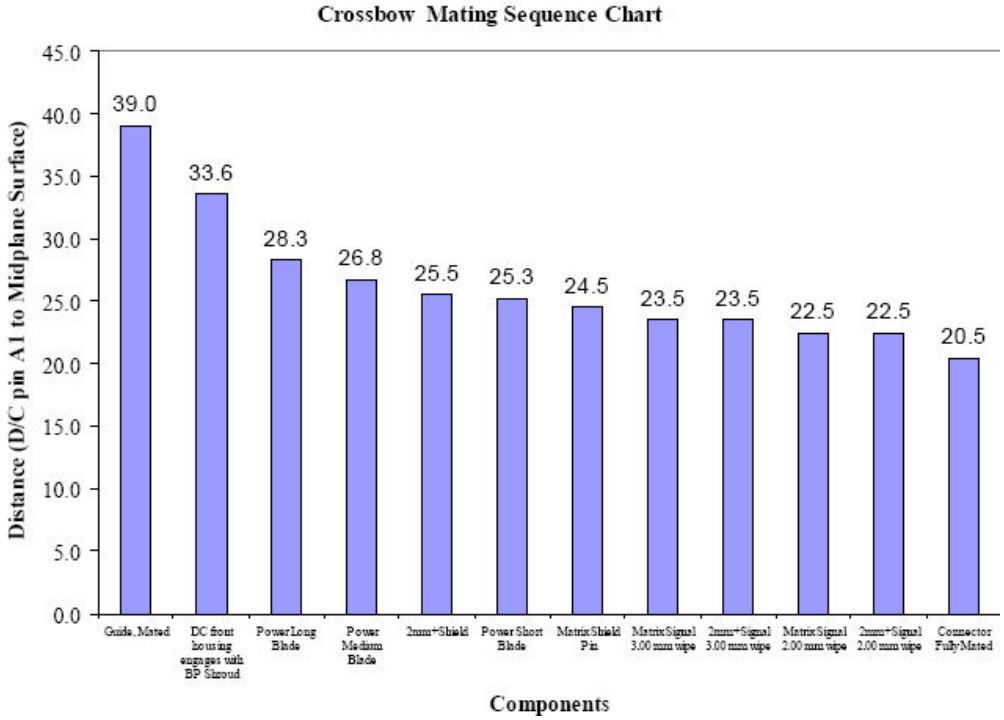


Figure 7: Mating Sequence and Wipe Distance

7.0 PCB DESIGN RULES AND MANUFACTURABILITY GUIDELINES

7.1 Footprint
For specific connector footprint, see customer drawings.

7.2 Copper Thickness
For copper thickness requirements and finished hole size reference, see table 2.

7.3 Minimum PCB Thickness
Recommended minimum PCB thickness of 0.063” (1.60mm).

7.4 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.

7.5 Hole Size and Finish
Reference Figure 8 & Table 2 for detailed dimensional recommendations regarding plated through holes.

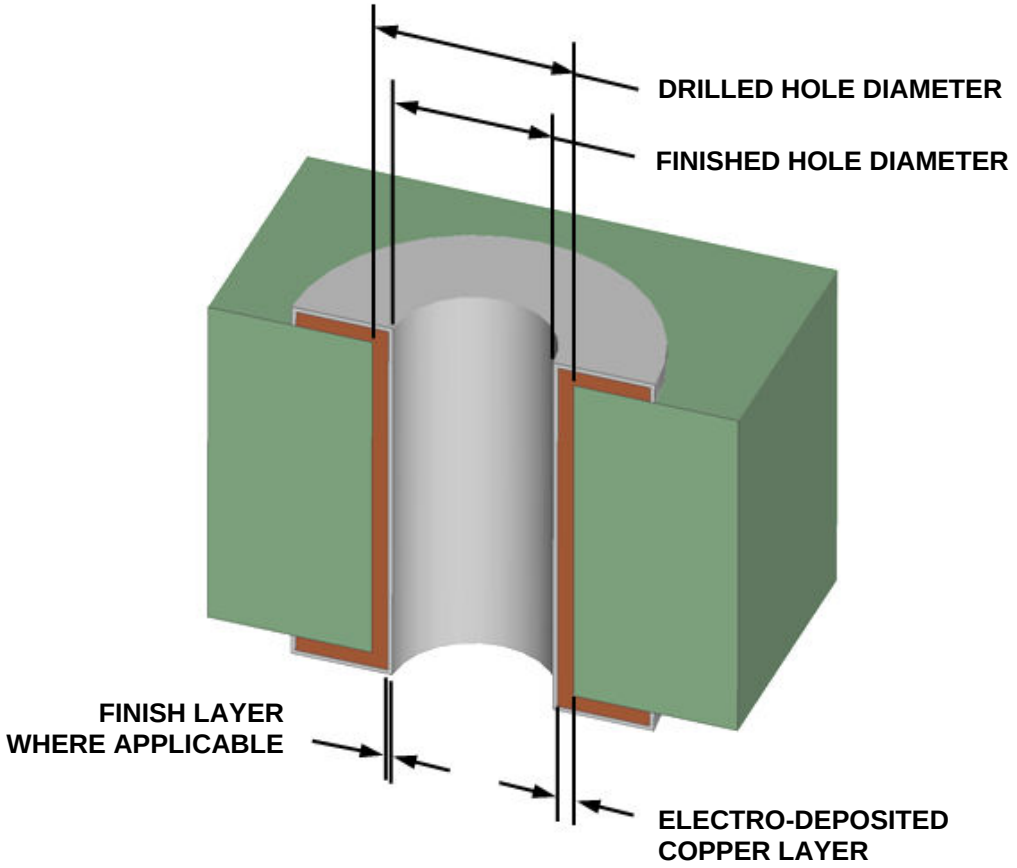


Figure 8: Crossbow™ Typical Finished Hole Cross-Section

	PTH Finish Type				
	Solder Finish (1)	**Immersion Sn (Tin)**	**Immersion Ag (Silver)**	**Copper - OSP**	**Ni Au (Nickel – Gold)**
Drill Size	ISO 0.57mm (.0225") Drill #74				
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	0.46±0.05mm (.0180±.0020")				

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

Table 2: Crossbow™ Plated Through-Hole (PTH) Requirements

7.6 Compliant Pin Critical Zone

There is a "Critical Working 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 "Non Critical Zone" area of the hole which is beyond the "Critical Working Zone", the plated through hole may go below the minimum specified hole size, (0.016"), *for non midplane applications*.

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

Please reference figure 9 for dimensional details regarding the compliant pin critical zone.

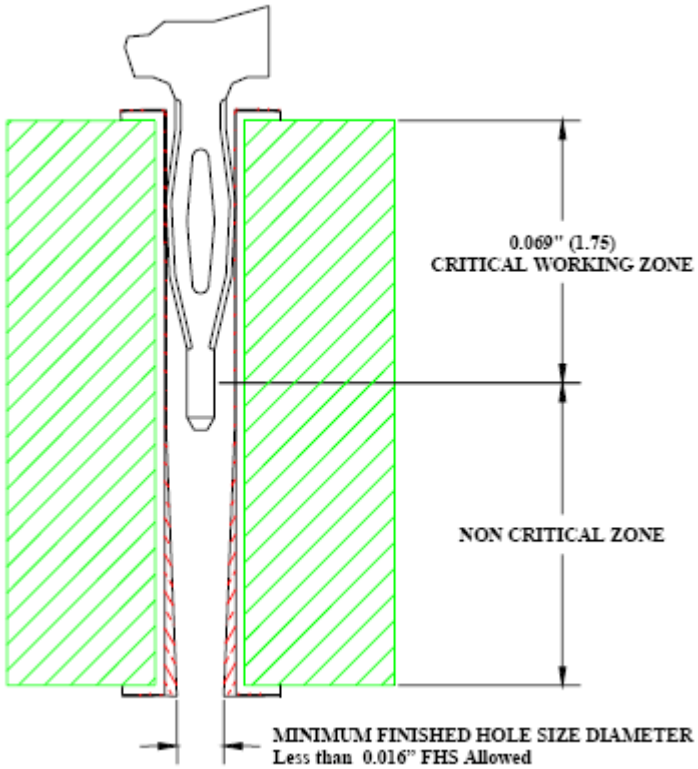


Figure 9: Crossbow™ Compliant Pin Critical Zone

8.0 PCB ROUTING GUIDELINES

8.1 Minimum Spacing
Minimum spacing, specific pad/trace, and trace/trace between all features should be 0.005” (0.127mm) to allow for manufacturing tolerances.

8.2 Impedance
Consider characteristic impedance, (if applicable), when designing to ensure line widths will meet requirements. Contact FCI Application Engineer for impedance calculations.

8.3 Copper Weights
Consider copper weights when routing. Higher weights will impact minimum trace widths.

8.4 Fillets
Fillets at the interface (egress) of the trace to the pad are required to improve annular ring when electrical design requires tight hole to pad configuration.

8.5 Trace Centering
Center all traces between holes to optimize spacing.

8.6 Drilled Hole and Copper Thickness

The specified drilled hole size and copper thickness is mandatory, see figure 8 and table 2.

8.7 Surface Traces

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

8.8 Line Widths, Pad Sizes and Spacing

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

8.9 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.

8.10 Non-functional Pads

Non-functioning pads on signal can be removed or added at the Designer's option.

8.11 Filleting

Filleting of pads is recommended (to be added by board fabrication house) for 0.000" annular ring (tangency), see Figure 10.

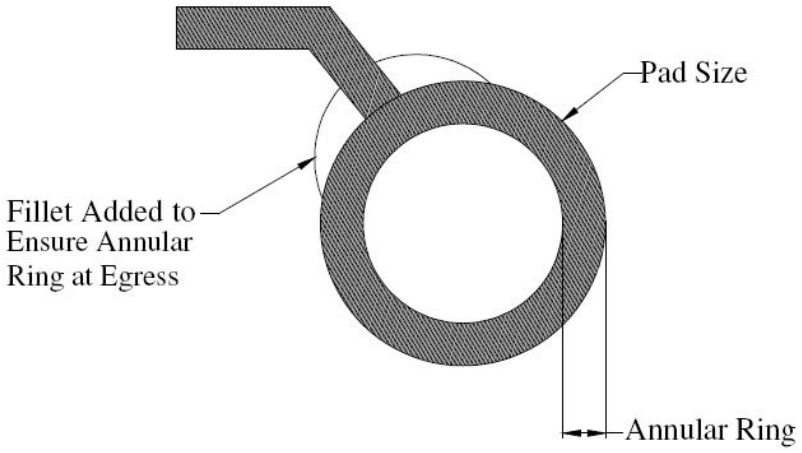


Figure 10: Preferred Fillet

Fillet diameter equals one half size of plated through hole pad diameter located on a line central to the trace so that the fillet size equals minimum annular ring plus 0.005" (0.13mm).

8.12 Crossbow™ Matrix Signal and Ground Pad Sizes

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 0.350 (1.60 to 8.80)	0.5 (17µm)	Inner Layer	.035 (0.89)	.037 (0.94)	.039 (0.99)
		Outer Layer	.0375 (0.95)	.0395 (1.00)	.0415 (1.05)
0.062 to 0.260 (1.60 to 6.60)	1.0 (35µm)	Inner Layer	.036 (0.91)	.038 (0.96)	.040 (1.02)
		Outer Layer	.0375 (0.95)	.0395 (1.00)	.0415 (1.05)

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 8.9. For all other plane clearances, contact FCI Applications Engineering.*

Table 3: Minimum Pad Size vs. Copper Weight and Annular Ring

8.13 Typical Anti-pad Geometry

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

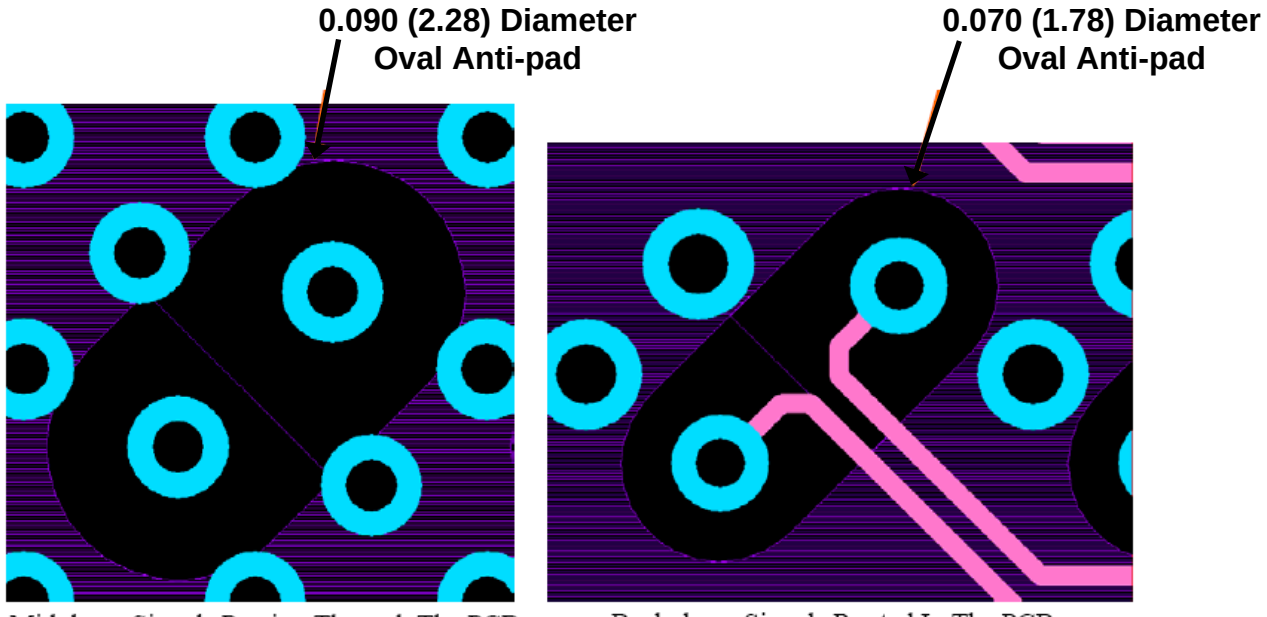


Figure 11: Anti-pad Clearance - Crossbow™ Matrix Backplane Connector

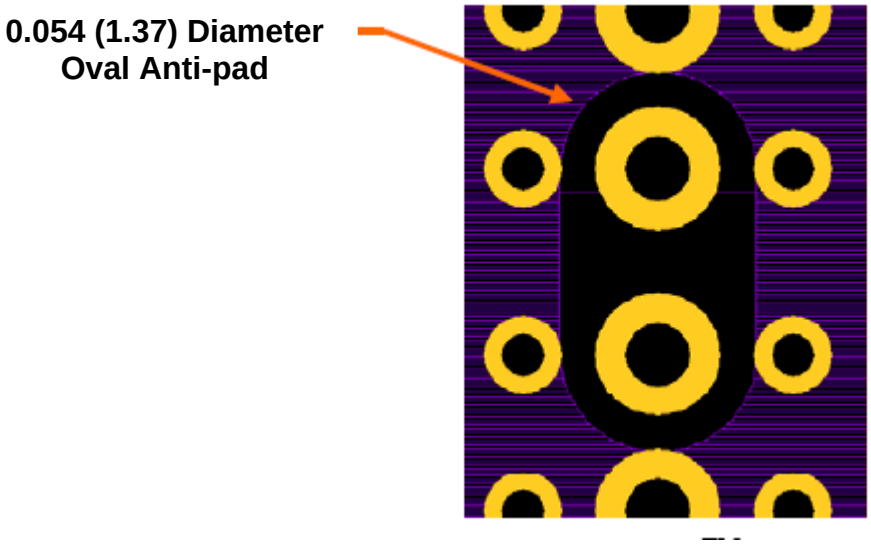


Figure 12: Anti-pad Clearance - Crossbow™ Matrix Daughtercard Connector

8.14 Crossbow™ Matrix Typical High-Speed Differential Routing

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

8.14.1 Crossbow™ Matrix Daughtercard Routing

8.14.1.1 Crossbow™ Matrix Daughtercard Routing With "Shadow Vias"

There are two dimensions that limit the amount of space available for routing the daughtercard footprint when shadow vias are used for additional electrical performance. They are the dimensions between ground vias and are labeled Dim "A" and "B" in figure 13. Dimension "A" and "B" do not change with differences in annular ring or copper weight. The following table details how to calculate the space available for routing between two ground holes on an inner layer, where there is no pad present.

Description	Routing Channel A	Routing Channel B
Center-to-Center	0.051" (1.300mm)	0.084" (2.120mm)
Drill Diameter	0.016" (0.400mm)	0.016" (0.400mm)
- 2* Space needed for drill wander	0.006" (0.152mm)	0.006" (0.152mm)
- 2* Clearance to the trace	0.010" (0.254mm)	0.010" (0.254mm)
= Resulting Space for Traces	**0.019" (0.482mm)**	**0.052" (1.320mm)**
Recommended Line Width	**0.006" (0.152mm)**	**Same as Dim "A"**
Recommended Space Between Lines	**0.007" (0.178mm)**	**Same as Dim "A"**

Table 4: Crossbow™ Matrix Daughtercard Routing Channels with Shadow Vias

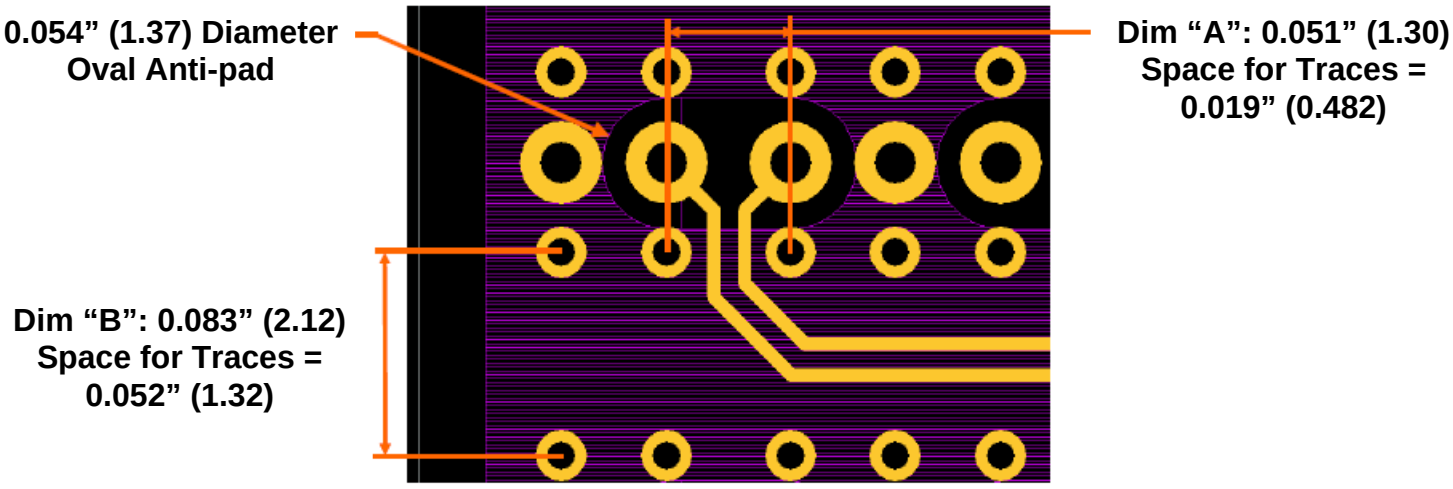


Figure 13: Crossbow™ Matrix Daughtercard High Speed Differential Routing With Shadow Vias

8.14.1.2 Crossbow™ Matrix Daughtercard Routing Without “Shadow Vias”

In this version of the footprint, the additional vias are removed to allow for more traces to be routed in the channel. The 6x6 matrix connector can be routed in 2 routing layers using this method. The available space for routing is determined by the distance between anti-pads, not annular ring or copper weight.

Description	Routing Channel A
Center-to-Center	0.157” (4.000mm)
Anti-Pad Diameter	0.054” (1.372mm)
= Resulting Space for Traces	**0.103” (2.616mm)**
Recommended Line Width	**0.006” (0.152mm)**
Recommended Space Between Lines	**0.007” (0.178mm)**
Recommended Space Between Diff Pairs	**0.020” (0.508mm)**
Resulting Ground Plane Web Overhang on Each Side	**0.003” (0.076mm)**

Table 5: Crossbow™ Matrix Daughtercard Routing Channels without Shadow Vias

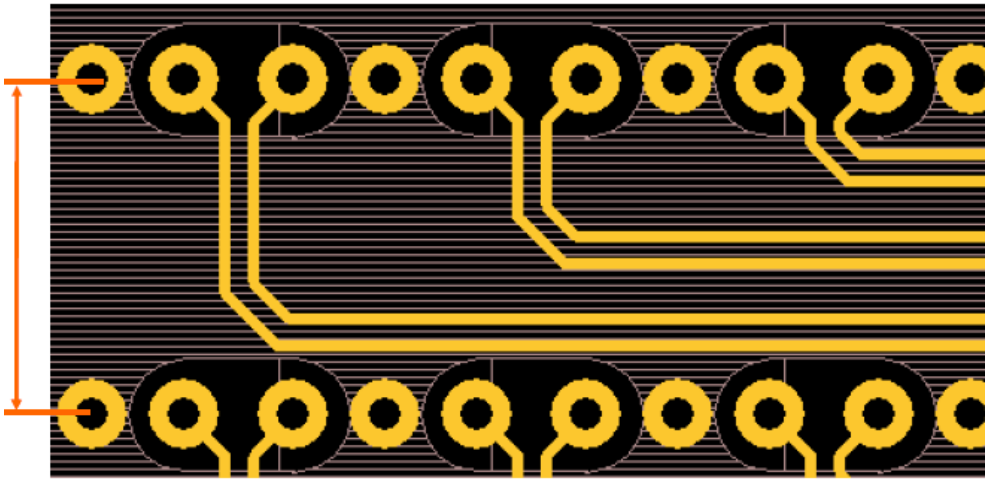


Figure 14: Crossbow™ Matrix Daughtercard High Speed Diff Routing Without Shadow Vias

8.14.2 Crossbow™ Matrix Backplane Routing

There are two backplane footprints for the Crossbow™ matrix backplane, one for midplane applications where signals are passing through the PCB and one for applications where traces are routed to another connector. This section will address the latter case.

The amount of space available for routing is determined by the ground plane web in between the anti-pad clearances. This routing space does not change with differences in annular ring or copper weight because the anti-pad size is independent of annular ring and copper weight.

Center-to-Center	0.105” (2.667mm)
Anti-Pad Diameter	0.070” (1.778mm)
= Resulting Space for Traces	**0.035” (0.889mm)**
Recommended Line Width	**0.007” (0.178mm)**
Recommended Space Between Lines	**0.008” (0.203mm)**
Resulting Ground Plane Web Overhang	**0.006” (0.165mm)**

Table 6: Crossbow™ Matrix Backplane Routing Channel

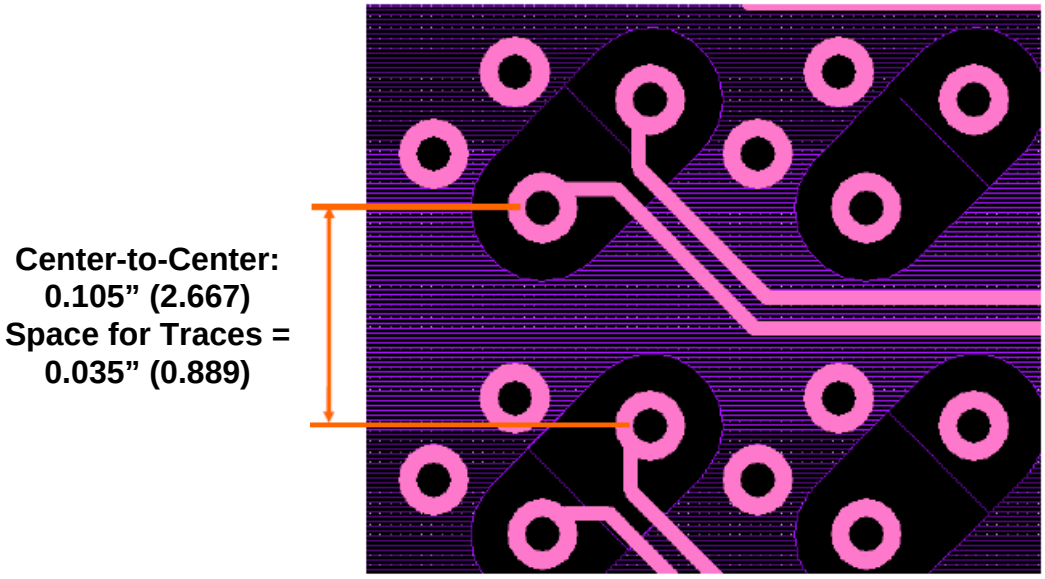


Figure 15: Crossbow™ Matrix Backplane Typical High Speed Differential Routing

9.0 INSTALLATION REQUIREMENTS

9.1 Application Press System Requirements

9.1.1 General

Crossbow press fit style components have been tested and utilized with many different press platforms. There are certain features and capabilities we recommend having. At a minimum the application press should have the following:

9.1.1.1 Force

The application press must have the capability of installing connectors with a maximum force of 35.6N (8lbs) per press-fit tail. Note: Do not exceed the maximum force of 8lbs/pin; damage to the PCB and the connector may result. Please refer to FCI product specification for specific performance criteria regarding press-fit tails.

9.1.1.2 Rate

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

9.1.1.3 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 25.4mm (1.00”) or greater. The press must have a Z-axis control to within 0.005” (0.127). The press bed must be parallel to the pressing surface of the insertion tooling. The face of the application press should be parallel with the PCB surface to within 0.005” (0.127).

9.1.1.4 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 222.4N (50 lbs) during the pressing cycle, and adjust the press cycle based on force feedback.

9.2 Application Tooling

9.2.1 Daughtercard Receptacle

The only application tools recommended for Crossbow™ Daughtercard Receptacles is a simple press-block as shown in figure 16. Crossbow application tools utilize features to locate and ensure the tool will support and apply forces as required for assembly. The tools are also designed to be stacked. 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 x 4	20mm / .79”	10101917-001
6 x 6	18mm / .71”	10101913-001
6 x 6	26mm / 1.02”	10101914-001
6 x 6	42mm / 1.65”	10101915-001
6 x 6	102mm / 4.02”	10101916-001

Table 7: 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.

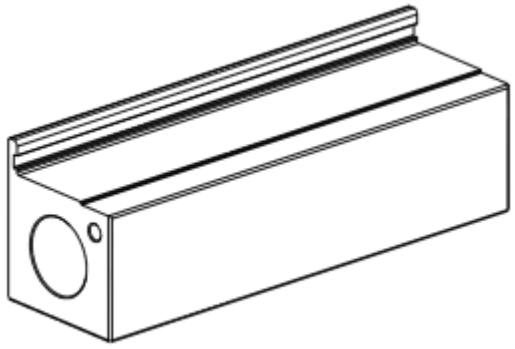
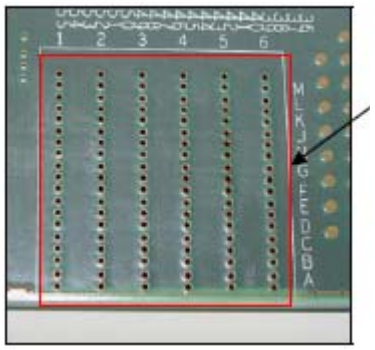


Figure 16: Crossbow™ Daughtercard Receptacle Press-Block

9.2.1.1 Printed Circuit Board Support

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 thickness of any PCB may vary by +/-10%. The selected material should allow minimal deflection under the maximum compression force. Please contact your FCI applications engineer for any needed assistance.



Matrix
4x4 0.65" (16.5mm) SQ
6x6 0.95" (24mm) SQ

Figure 17: Minimum Recommended Support Area for Matrix 4x4 and 6x6

9.2.2 Backplane Header

The only application tools recommended for Crossbow™ Backplane Headers are custom press-tools and support tools shown in figures 18 thru 23. There are specific tools required for each application. Please refer to the following table for sizes and part numbers of this tooling.

Press-Tool Description		Press-Tool Part Number
Insertion Tool for 4x4, Vertical Header	(Fig 18)	10101920-001
Insertion Tool for 6x6, Vertical Header	(Fig 19)	10101921-001
Bottom Loading Head, U base, 90 Deg, 4x4	(Fig 20)	10101923-001
Bottom Loading Head, Flat, Universal, 4x4	(Fig 21)	10101924-001
Bottom Loading Head, U base, 90 Deg, 6x6	(Fig 22)	10101925-001
Bottom Loading Head, Flat, Universal, 6x6	(Fig 23)	10101926-001

Table 8: Crossbow™ Backplane Header Press-Tool Part Numbers

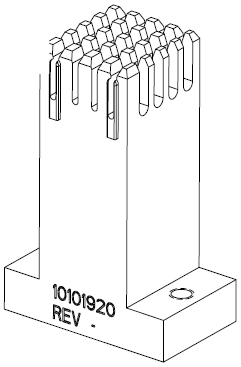


Figure 18:
4x4 Insertion Tool

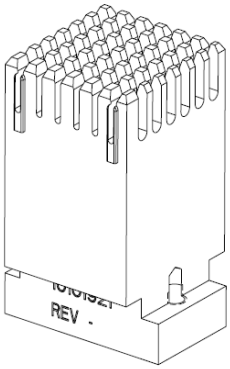


Figure 19:
6x6 Insertion Tool

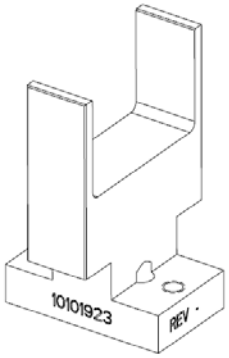


Figure 20:
4x4 U-Base
Bottom Loading Head

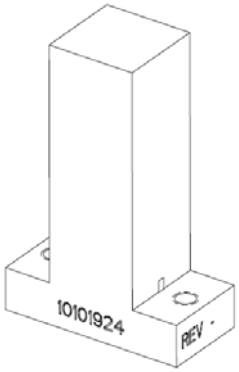


Figure 21:
4x4 Flat Universal
Bottom Loading Head

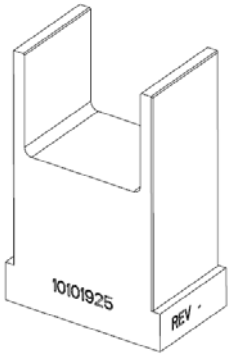


Figure 22:
6x6 U-Base
Bottom Loading Head

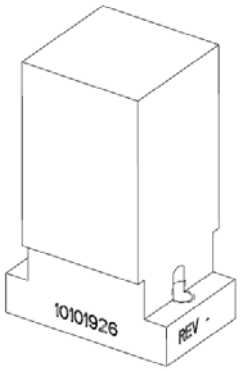


Figure 23:
4x4 Flat Universal
Bottom Loading Head

9.2.2.1 BP Insertion Head

Insertion heads are inserted into Crossbow™ backplane connectors after the connector is aligned on the PCB for pressing and are designed to support and withstand forces required for assembly.

9.2.2.2 Printed Circuit Board Support

Application of the backplane vertical header requires an appropriate backup fixture to support the PCB during the connector installation process. The configurations of the insertion base tools provide adequate support directly under the connector assembly based on the footprint pattern. There are two types of insertion base tools for each application of Crossbow™ backplane connectors; flat base and U-shaped base. The U-shaped base is only used in instances when there is another connector already pressed into the same location on the opposite side of the PCB or if the PCB is thinner than 2.1mm, (thinner than the length of the compliant pin). 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 an FCI applications engineer for assistance.

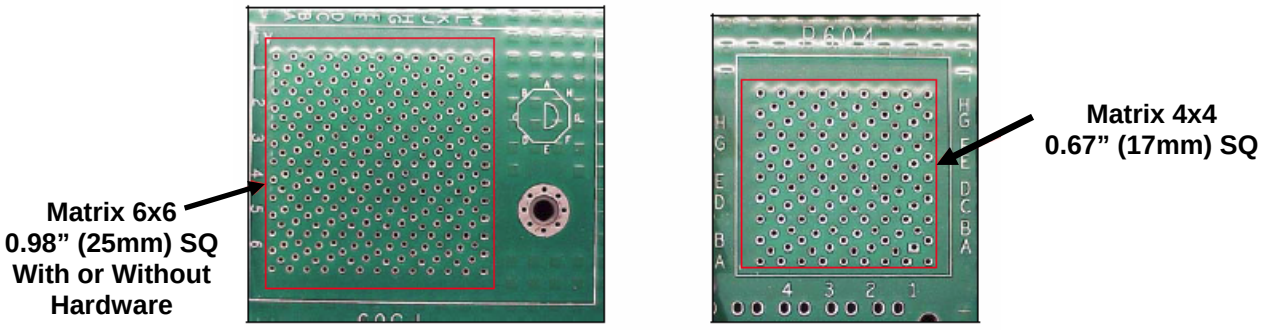


Figure 24: Minimum Recommended Support Area for 6x6 and 4x4 BP Connectors

9.2.2.3 Support Limitations

There are circumstances where it is not feasible to provide a support area that encompasses the entire connector pattern, i.e. interference with SMT components, etc. In cases such as these, consult with the board manufacturer regarding minimum support area for the specific application.

9.3 Application Procedure: Daughtercard Receptacle

9.3.1 Alignment of the Product

The application tool is to be placed on the connector and not attached to the press ram. Therefore the alignment of the PCB in the press is such that the press ram is approximately, (within a diameter of 0.5”), centered above the tool.

9.3.2 Press Cycle

The application tool is designed to conform to the stiffener profile. Place tool on connector and lower the press head until the head reaches the application tool. Continue the press cycle until the connector is seated flush on the PCB surface.

Figure 25: Crossbow™ DC matrix 4x4 Prepped for Seating with Application Tool

9.4 Inspection Procedure: Daughtercard Receptacle

9.4.1 Position

Verify the final seating depth of the connector, which must be flush to the PCB or less than 0.005" (0.13mm) above the surface of the board. A gap of 0.005" (0.13mm) or greater between the PCB and the connector is NOT acceptable.

9.4.2 Quality

Verify the final seating depth of the connector, using a 0.005" (0.13mm) shim. The shim should not fit between the surface of the PCB and the plastic stand-offs of the connector.

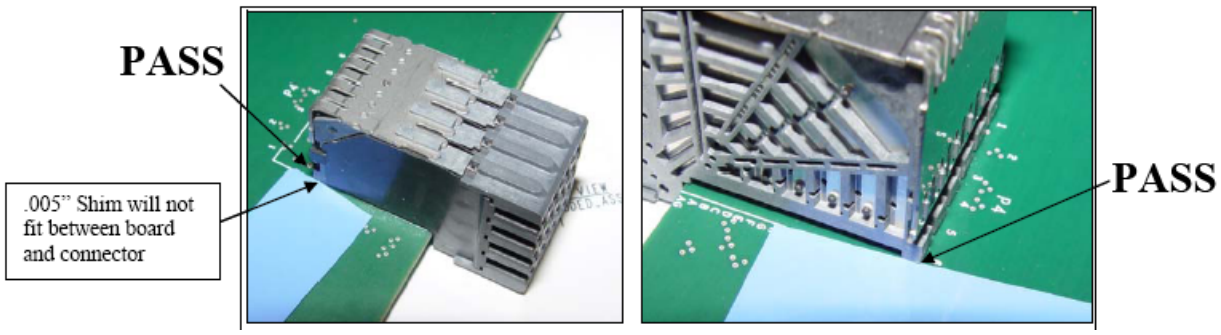


Figure 26: Acceptable Condition as Tested with a 0.005" (0.13) Shim

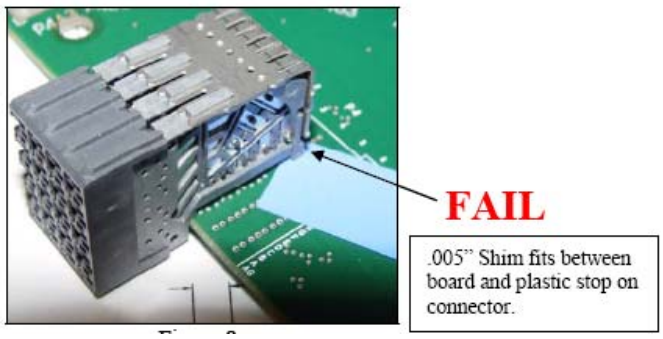


Figure 27: Unacceptable Condition as Tested with a 0.005" (0.13) Shim

Verify that there are no cracks or deformities in the plastic.

If possible view all compliant pins from the backside of the PCB. All compliant pins must be visible and be pressed to the same Z position.

9.5 Application Procedure: Backplane Header

9.5.1 Alignment of the Product

The application tool (insertion tool) can be manually placed on the mating side of the connector or attached to the press ram. The alignment of the PCB in the press is such that the press ram is approximately, (within a diameter of 0.5”), centered above the tool.

9.5.2 Press Cycle

The application tool (insertion tool) is designed to conform to the mating end geometry of the connector. Ensure that the connector is properly pre-loaded, place the PCB into the press and place the insertion head on the connector. Lower the press head until the head reaches the insertion head. Continue the press cycle until the connector is seated flush on the PCB surface.

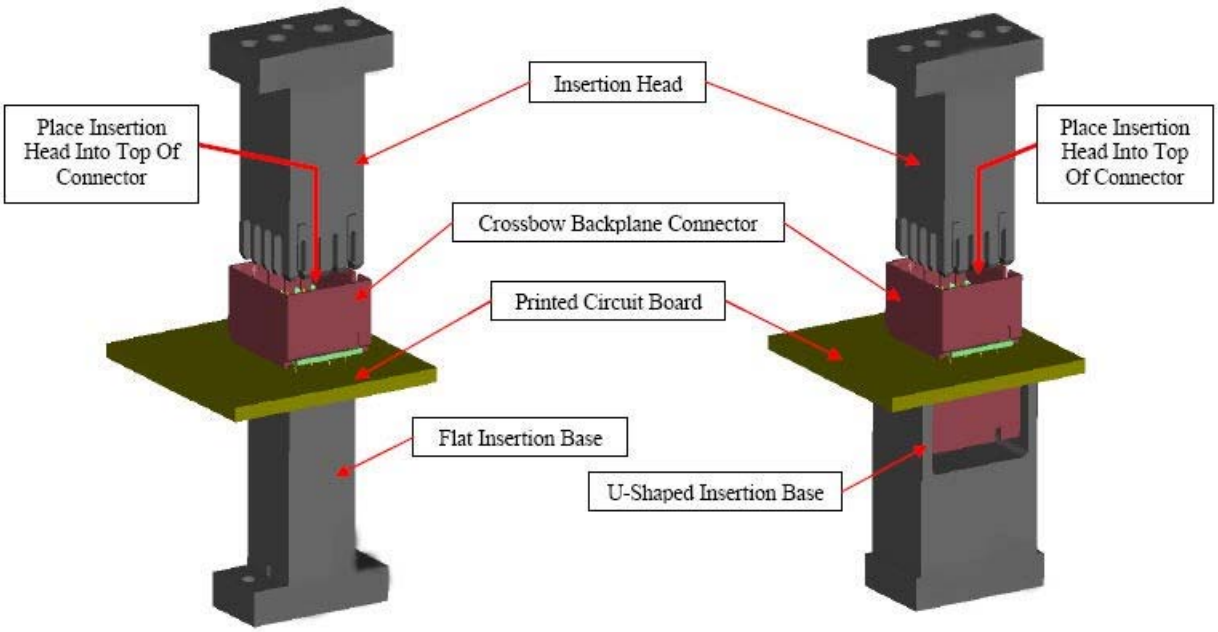


Figure 28:
Setup using Flat Universal Bottom Loading Head

Figure 29:
Setup using U-Base Bottom Loading Head

9.6 Inspection Procedure: Backplane Header

9.6.1 Position

Verify the final seating depth of the connector, which must be flush to the PCB or less than 0.005” (0.13mm) above the surface of the board. A gap of 0.005” (0.13mm) or greater between the PCB and the connector is NOT acceptable.

9.6.2 Quality

Verify the final seating depth of the connector, using a 0.005" (0.13mm) shim. The shim should not fit between the surface of the PCB and the plastic stand-offs of the connector.

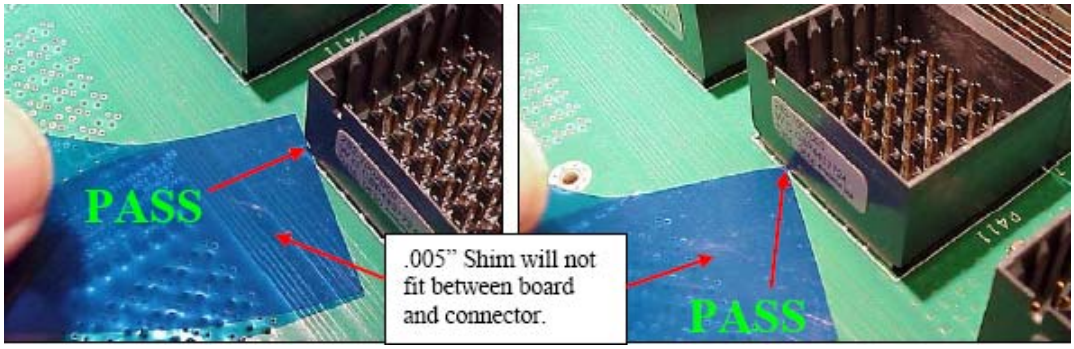


Figure 30: Acceptable Condition as Tested with a 0.005" (0.13) Shim

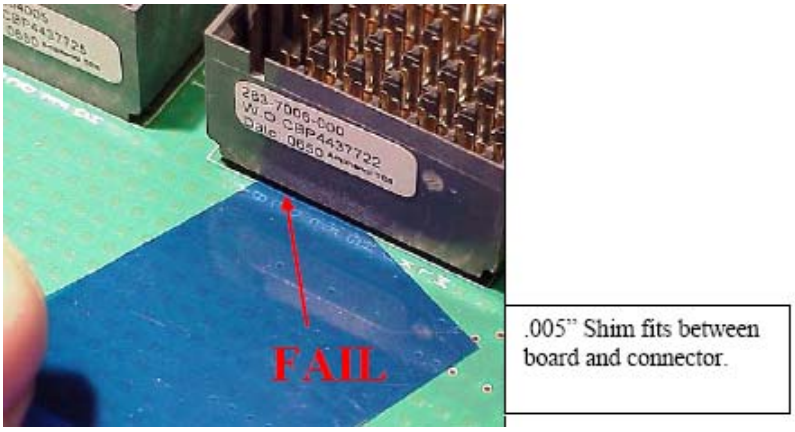


Figure 31: Unacceptable Condition as Tested with a 0.005" (0.13) Shim

Verify that there are no cracks or deformities in the plastic.

If possible view all compliant pins from the backside of the PCB. All compliant pins must be visible and be pressed to the same Z position.

10.0 REVISION RECORD

REV	PAGE	DESCRIPTION	EC #	DATE
1	all	Initial draft	-	2010-01-18