Table of Contents

Section **Page No.**

1.0 OBJECTIVE

This specification provides information and requirements regarding customer application of the ExaMEZZ connector systems with press-fit termination. This specification is intended to provide general guidance for application process development. It is recognized that no single application process will work under all customer scenarios and that customers will develop their own application processes to meet their needs. However, if these application processes differ greatly from the one recommended, FCI cannot guarantee acceptable results.

2.0 SCOPE

This specification provides information and requirements regarding customer application of ExaMEZZ connector systems with press-fit termination.

3.0 GENERAL

This document is meant to be an application guide. If there is a conflict between the product drawings and specifications, the drawings take precedence.

3.1 Product Description and Features

The ExaMEZZ connector systems provide high speed performance. The ExaMEZZ connector utilizes a hermaphroditic assembly (plug and receptacle in one) for use with mezzanine-type applications. The product is keyed to ensure proper mating orientation. The ExaMEZZ connectors use compliant press-fit tails to provide a reliable electrical connection between the connector and the plated through hole (PTH) of the PCB. The ExaMEZZ products have a small press-fit section, designed for a Ø0.36mm finished through hole, for signal terminals and a standard press-fit section, designed for a Ø0.50mm finished through hole, for the ground terminals. In addition to the signal pairs and grounds, there is a single low speed signal on the end of each column that may be utilized for miscellaneous low speed signals and/or low power requirements.

3.2 Product Configurations

The ExaMEZZ Hermaphroditic connector is currently offered as shown in Table 1 below.

Stack Height per Connector Half *	Signal Pairs per Column	Number of Columns	Column Spacing	Number of Positions (Including Grounds)
7.5mm	2 Pair	15, 19	2.0mm	120, 152
12.5mm	2 Pair	15	2.0mm	120
19.5mm	2 Pair	15, 19	2.0mm	120, 152
22.5mm	2 Pair	19	2.0mm	152
19.5mm	4 Pair	11	2.0mm	154
19.5mm	4 Pair	25	2.0mm	350

Table 1: ExaMEZZ Hermaphroditic Connector Configuration Offerings*

* For example, a 7.5mm connector mated to a 19.5mm connector results in a 27.0mm stack height board to board. The mated stack height tolerance between boards is approximately ±0.25mm.

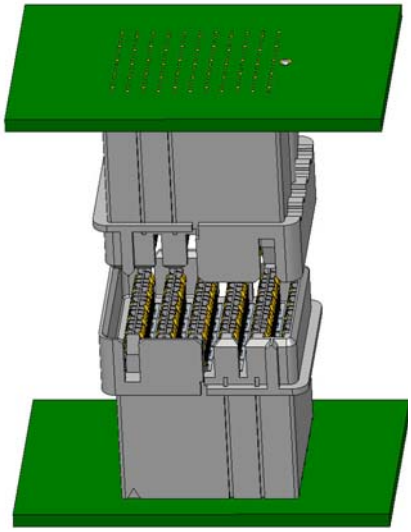


Figure 1: ExaMEZZ Hermaphroditic Connector Assembly (4 Pair, 2mm Pitch, 11 column shown)

3.3 Connector Guiding Features

Table 2 below shows the amount of misalignment in each direction that can be tolerated by the guiding features on the connector housings as they are mated.

Connector type	Nominal misalignment correction in direction parallel to connector columns	Nominal misalignment correction in direction perpendicular to connector columns
ExaMEZZ, all types	± 1.4 mm	± 1.4 mm

Table 2: Connector Guidance Features

3.4 PCB Spacers/Standoffs

Proper system application of ExaMEZZ Connectors requires the use of mechanical spacers/standoffs. The purpose of these spacers is to provide strain relief of connectors and connector intermate retention.

- Spacer height should be the nominal mated connector board-to-board height +0.10/-0.00mm.
- Holes on bottom PCB should be close fit to bottom screw major diameter. To assure proper mating of connectors, the top PCB holes can be up to 1.0 mm larger than spacer through shoulder diameter.
- The through shoulder can be a cylinder with internal thread or an external thread, as shown below.
- A minimum of 4 spacers around the connector is recommended.

• Spacers are not provided with connectors. A typical spacer is shown in Figure 2 and can be purchased from hardware vendors.

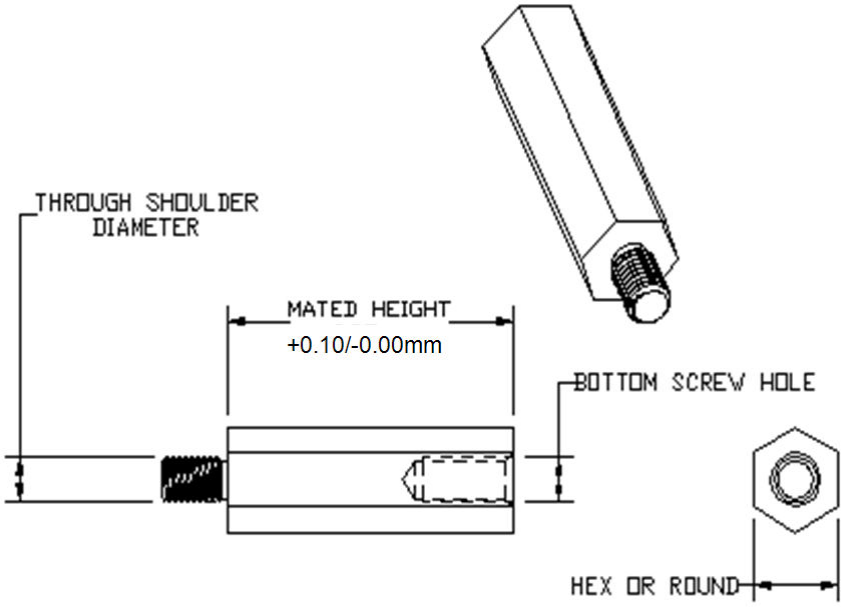


Figure 2: PCB Spacers

3.5 Mating / Unmating Methods

Whenever possible, the ExaMEZZ connector system should be mated straight on with PCBs near parallel. However, when there is a large contact count connector or multiple connectors on the board, it may be advantageous to mate the connector on a slight angle as depicted in the next few figures below to lower total mating force.

- First Pre-Mate Connectors near parallel such that keys and center datum X wall are engaged. This will allow connectors to best align themselves. Beams will start to engage at a position slightly beyond this location.

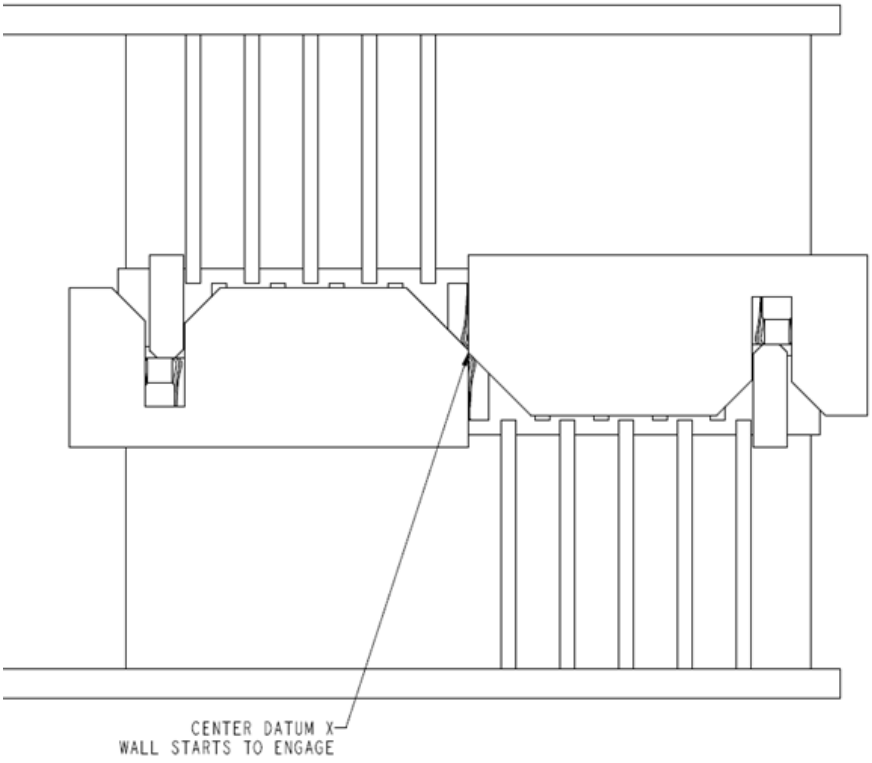


Figure 3: Pre-mate Location

- Slightly push down one side of the connector to a maximum of 1.5°. The example below shows the right hand side, but the left hand side could be chosen as well. Contacts will start to engage.

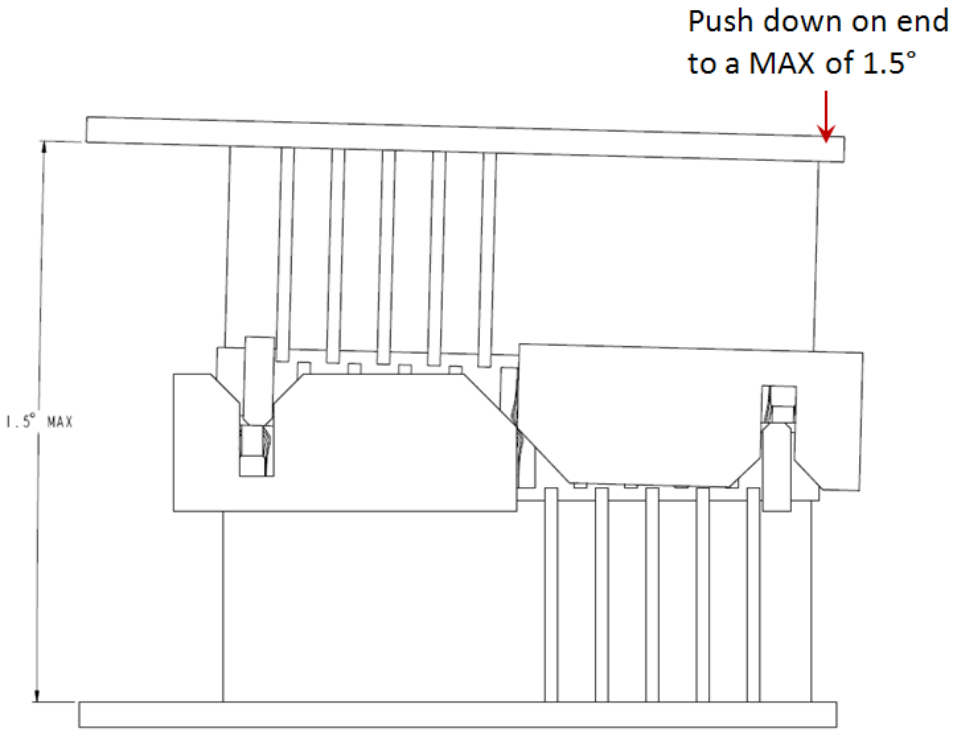


Figure 4: Slightly Push One End to 1.5º Maximum

- Continue Mating at 1.5º Maximum until near the bottom.

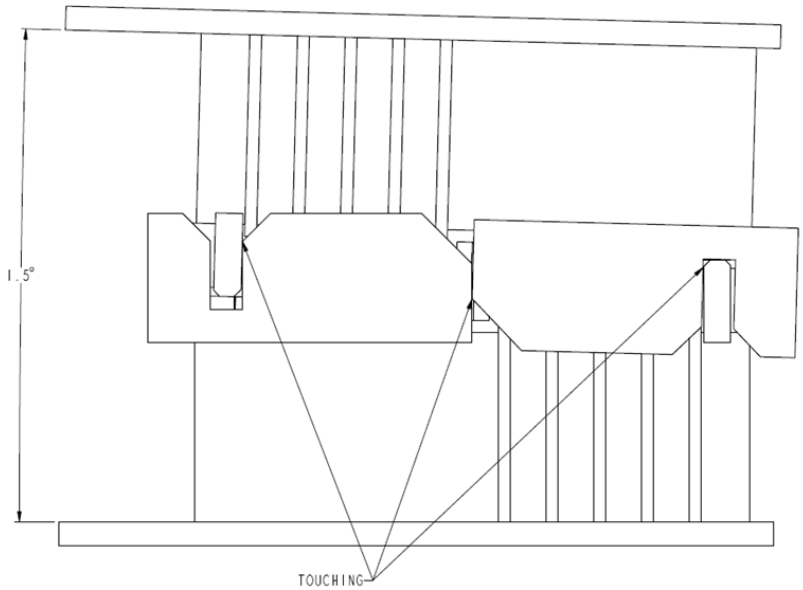


Figure 5: Continue Mate at 1.5º Maximum until Bottoms

- When bottom is reached, ensure connectors and PCB's are brought back close to parallel

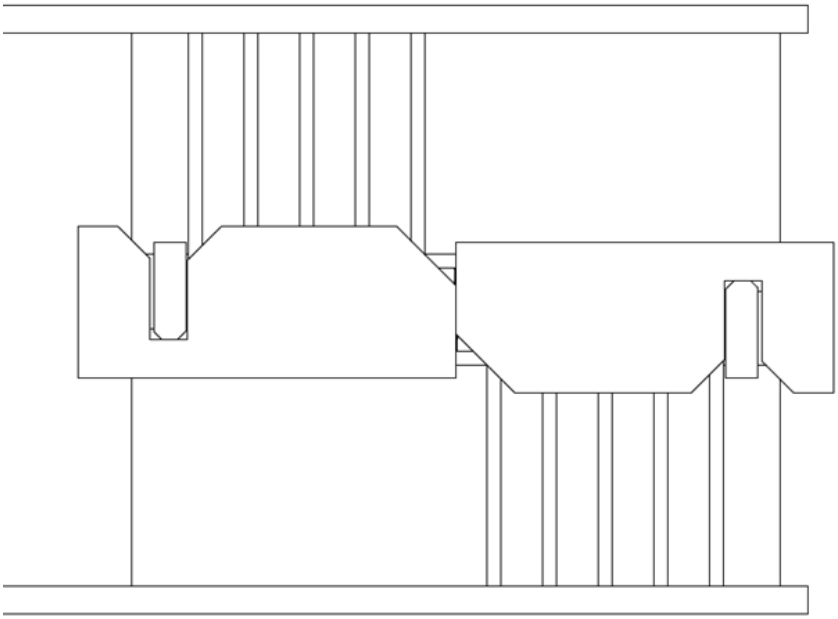


Figure 6: Final Mate Condition - Near Parallel

Like mating the connector, the connector it is preferable to unmate the connectors by pulling them straight apart. However, it requires less effort if the connectors are rocked side to side with a maximum of 2° while pulling the boards apart as shown in the figure below.

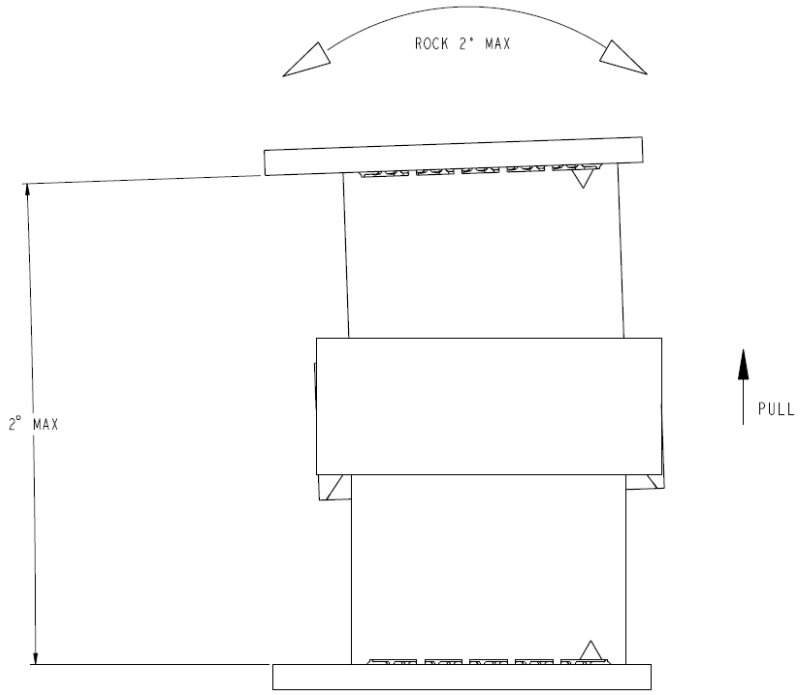


Figure 7: Unmate Rock Side to Side

3.6 Contact Mating and Connector Mating Sequence

The ExaMEZZ connector system is capable of providing three different mating wipe lengths. When specifying the short detect configuration, only one connector side can have a short detect contact. When specifying the advanced mate configuration, both mating connectors need to have advanced mate contacts. See Tables 3 and 4 as well as the customer drawings for additional details.

Item	Standard Mate Only	Advanced Mate	Short Detect
Connector Suffix	-XXXXX0ULF	Not standard (see customer drawing)	-XXXXX1ULF
Contact Location	All	2nd and 2nd Last Column, Top Contact	First and Last Column, Row A

Table 3: Contact Mating Details

Un-Mate Nominal Distance (mm)	Description
6.0	Housing Keys and Keyways start to align connectors
4.7	Housing Keys and Keyways are fully engaged
4.4	Advanced Mate contacts first touch
3.7	Advanced mate contacts reliable wipe begins
3.6	Standard Mate contacts first touch
2.6	Short Detect beam makes contact with Standard Mate beam
2.5	Standard Mate contacts reliable wipe begins
2.2	Short Detect contact reliable wipe begins
2.0	Standard Mate contacts begin two points of contact
0.0	Connectors fully mated

Table 4: Connector Mating Sequence

3.7 Power Options

The ExaMEZZ connector system can be configured to provide both high speed signal and power within the same connector. While any contacts within the ExaMEZZ connector can be used for power, we do not recommend using short detect contacts. If ground contacts are used for power, all the ground contacts within a column must be used for power because they are commoned together. Refer to ExaMAX ExaMEZZ Power Design Guide, GS-20-0456, for additional details.

4.0 PRINTED CIRCUIT BOARD LAYOUT INFORMATION

4.1 Board Thickness

The minimum PCB thickness that does not require PCB back-up tooling for EON tip clearance is 1.75mm. If back-up tooling will be used, the minimum PCB thickness is 1.30mm. There is no maximum thickness requirement.

4.2 Multiple Connector Applications and Spacing

The ExaMEZZ connectors in general are not stacked directly next to each other. Spacing will depend upon orientation to the board. Please refer to customer drawings for overall size, and allow at least 0.10mm clearance all around each mated connector pair for tolerance stack-up. This will result in a 0.20mm minimum gap between two ExaMEZZ connectors, see Figure 8 below. True position between patterns of PCB holes must be within Ø0.10mm (±0.05mm), and all connectors should have the same orientation; example is shown in Figure 9 below.

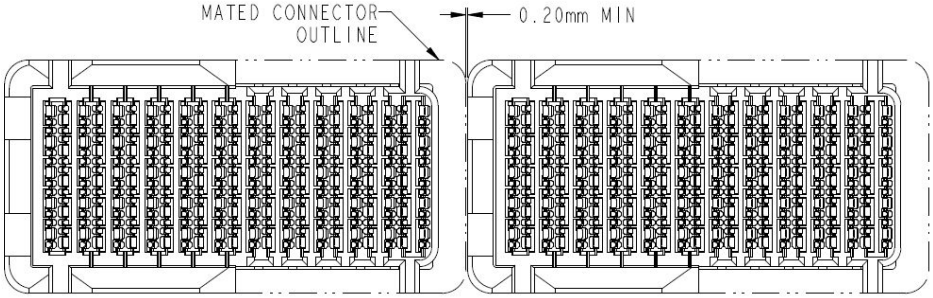


Figure 8: Minimum Spacing

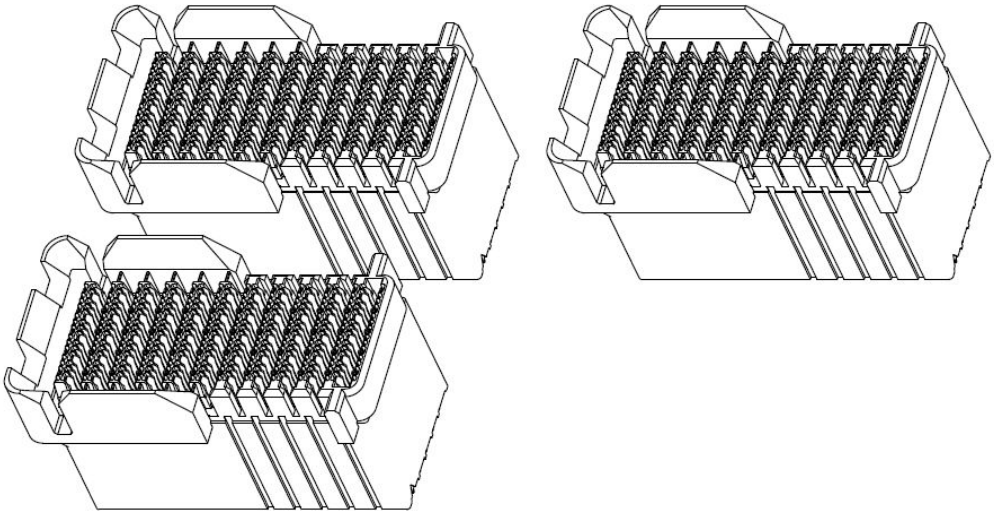


Figure 9: Oriented in Same Direction

4.3 Printed Circuit Board Screen Printing Recommendations

It is recommended to print the outline of the connector onto the PCB to insure proper placement of the connector. Detailed information of the recommended outline is provided on the customer drawing.

4.4 Keep-Out Zones for Application and Removal Tooling

There are no keep-out zones necessary for insertion application tooling because these tools fit within the outside envelope of the connector assemblies.

In order to minimize any strain on the board and other components near the connector, it is recommended that for PCB / connector rework, allowances on the connector side of the board should be made for 2mm keep-out zones adjacent to the connector on a minimum of two opposing sides. Support on all 4 sides of the connector is recommended. Refer to sections 5.7-5.8 for information on removal tools and procedures. In general, the need for keep-outs will depend on the specific PCB layout.

4.5 Row A Identification and PCB footprint labeling

Because ExaMEZZ hermaphroditic connectors are self-mating, it is important to understand position identification on the connector and the PCB footprint layout.

If this hermaphroditic connector was defined with a single position A1, then the A1 positions would not line up on the mated connectors. For an 11-column connector, A1 would align with position A11, etc. To eliminate confusion, there is a Row A indicator mark at each end of the connector as shown in Figure 10. When mated, the Row A indicator marks of the first connector must line up with the row A indicator marks of the second connector.

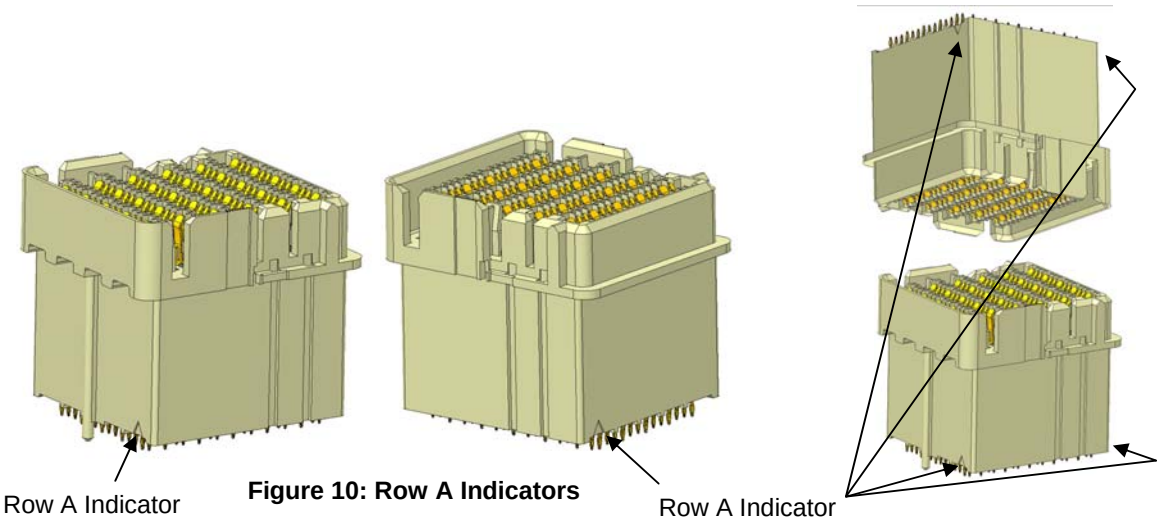


Figure 10: Row A Indicators

PCB #1 (usually the main system board) will number the columns from left to right. In order to align A1 positions of the mated boards, **PCB #2 (usually the mezzanine card)** will have the same footprint, but the column numbers will be a mirror image (right to left) of the PCB#1 layout. Please refer to Figure 11 and the applicable customer drawing.

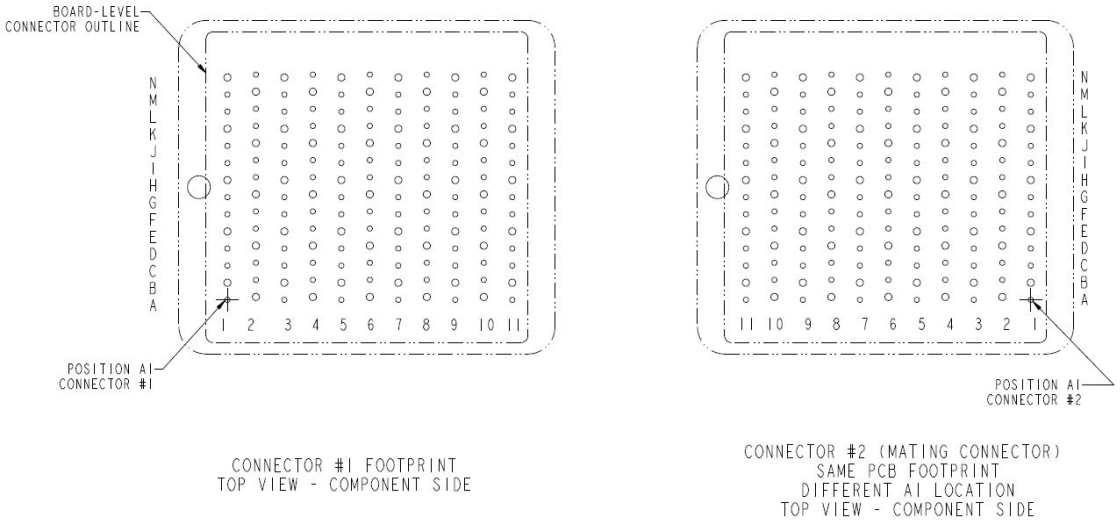


Figure 11: PCB footprint labeling

4.6 Column Offset

When the connector on PCB #2 is flipped over to mate with the connector on PCB #1, there is a column offset of 0.4mm between the PCB footprints as shown in Figure 12.

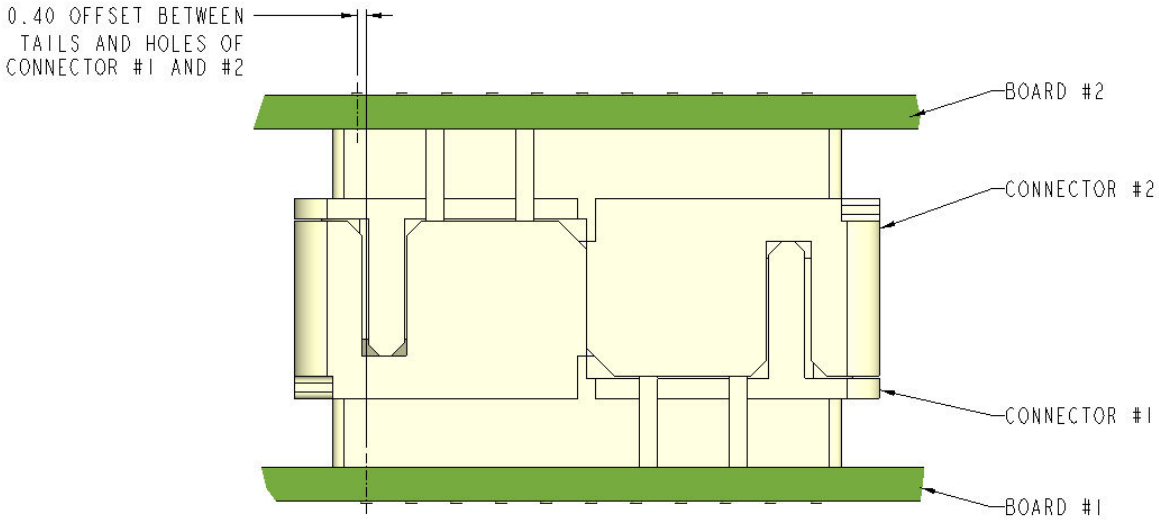


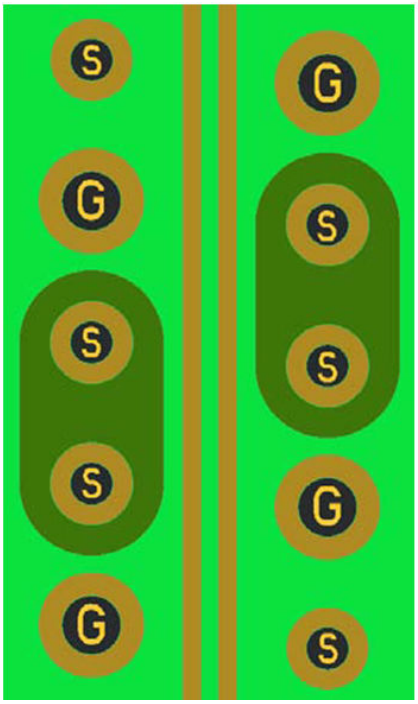
Figure 12: Mated PCB hole offset

4.7 Printed Circuit Board Trace Routing

To minimize trace loss in high-speed systems, board designers typically prefer to use trace widths no smaller than 0.10mm. The desired impedance between adjacent traces and the PCB layer thickness will define the spacing between those traces. The board designer will ultimately determine these parameters for a given system.

2.0mm Column Pitch

The routing channel is defined by the size of the anti-pads and the column spacing. One differential pair can be routed per layer using a 2mm column pitch. Refer to Figure 13 for a routing example. **Please note that the separation between traces and anti-pads is key to minimizing crosstalk between layers.**



	mils	mm
trace width	6.3	0.1600
trace-to-trace separation	5.7	0.1452
trace-to-antipad separation	6.0	0.1524
signal via(S) drill Ø	17.7	0.45
signal via(S) finish Ø	14	0.36
Signal via (S) pad Ø	27.7	0.70
ground via(G) drill Ø	23.6	0.60
ground via(G) finish Ø	20	0.50
antipad width	48.4	1.23
antipad length	110.2	2.80
row pitch		1.20
column pitch		2.00

Figure 13: PCB Trace Routing Example, 2mm Column Pitch

4.8 Trace breakout for mezzanine applications

Because traces can exit 2 sides of a mezzanine connector, the total number of routing layers can be as small as the number of pairs per column divided by 2. Therefore a 4 pair ExaMEZZ connector could be routed on 2 routing layers. Please see Figure 14.

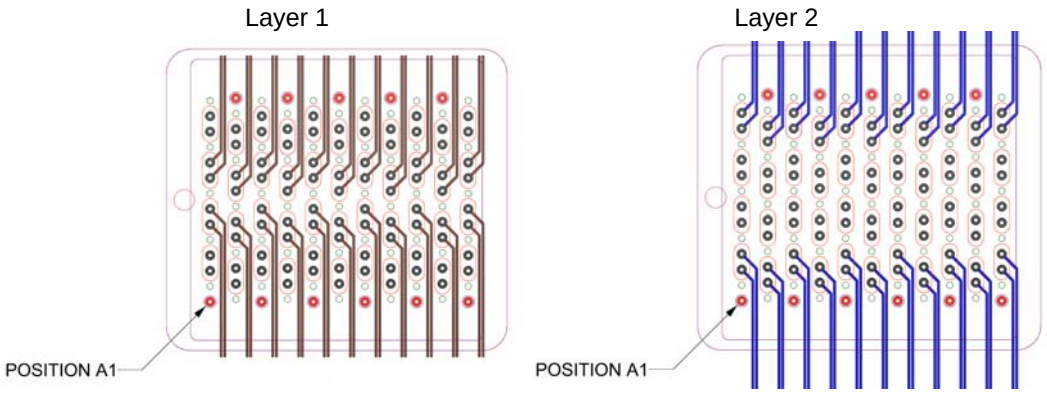


Figure 14: Trace Breakout Example

4.9 Low speed positions

Each column of an ExaMEZZ connector has an extra pin for low speed signal or other miscellaneous function. The finished hole size is the same as for all high speed signal positions

(0.36mm). The drill diameter and pad size recommendations from the differential layout will be the same for these locations. A 0.95mm diameter anti-pad is shown, but any size that prevents shorting to the ground layer will work since impedance is typically not important. Trace width is flexible also, depending on the function of the pin.

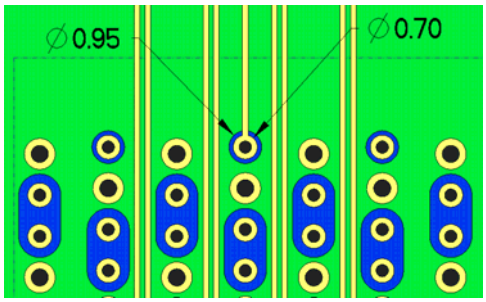


Figure 15: Low Speed End Signal PCB Finished Hole/Antipad Design

4.10 Skew compensation at connections to signal vias

When a pair of traces is connected to a pair of signal vias, the length of those traces will be different unless precautions are made. Any difference in length will result in intra-pair skew. Please see below for a method to minimize the difference in length. The image on the left shows the signal routing layer, while the image on the right shows the ground plane layer. The trace connection to the right signal via in each pair is made longer than the connection to the left via to account for overall length differences. Also, on the ground layer, there are extensions to the ground plane that are found beneath the signal traces as they extend into the anti-pads. These ground layer extensions will help to maintain the desired impedance for that length of trace. Without the ground beneath the trace, the impedance will be higher.

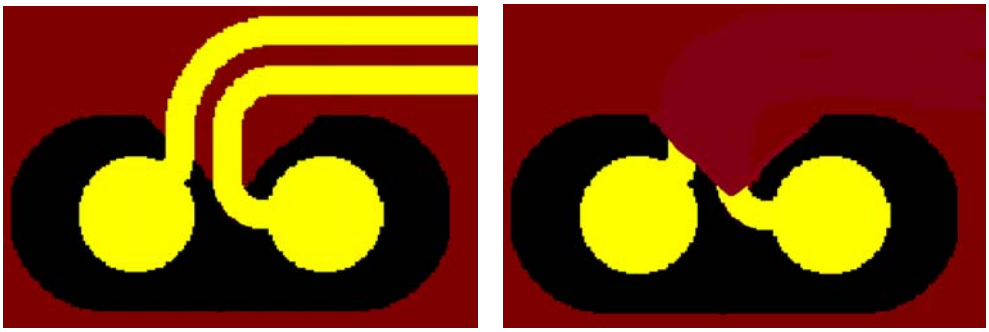


Figure 16: Skew Compensation

4.11 Back drilling

Back drilling is a method used by system designers that reduces the length of a conductive via which will improve high speed signal integrity performance of a link. When back drilling is performed it is important to avoid damaging the portion of the via that makes contact with any press-fit tails. The diameter of the back-drill should be at least 0.2mm (8 mils) larger than the drill size used for the plated via. See below for recommendations on proper back drilling.

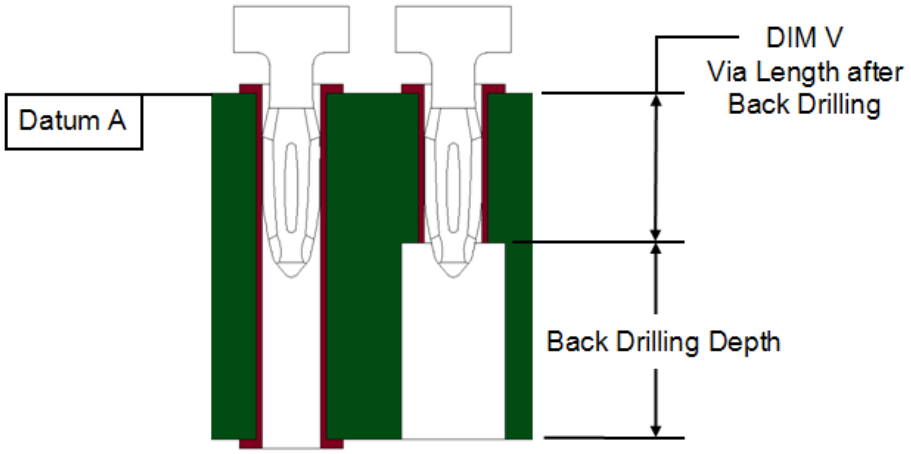


Figure 17: View of Back Drilling

After a back drilling operation, the remaining via barrel length (Dim V) must be at least 1.0mm for the signal terminals to ensure a reliable connection between the press-fit tails and the PCB. Assuming that the PCB manufacturer can maintain a back drilling depth tolerance of ± 0.3mm relative to datum A, the nominal via length after back drilling would need to be 1.3mm.

4.12 Anti-pads in back drilled regions

In regions where vias will be back-drilled, the anti-pad should be reduced to a diameter equal to the signal via drill hole + 0.4mm (16 mils). This assumes the back-drill size used is equal to the signal via drill + 0.2mm and the anti-pad is 0.2mm (8 mils) larger than the back-drill diameter to avoid contact between the drill and the ground layers being drilled through. These small anti-pads should only be used on non-reference ground layers in the back-drilled region. The ground plane immediately below the routed signal layer should still have the standard anti-pads for impedance matching. The smaller anti-pads will minimize layer-to-layer crosstalk leakage between any traces sharing the same routing channel on adjacent signal layers.

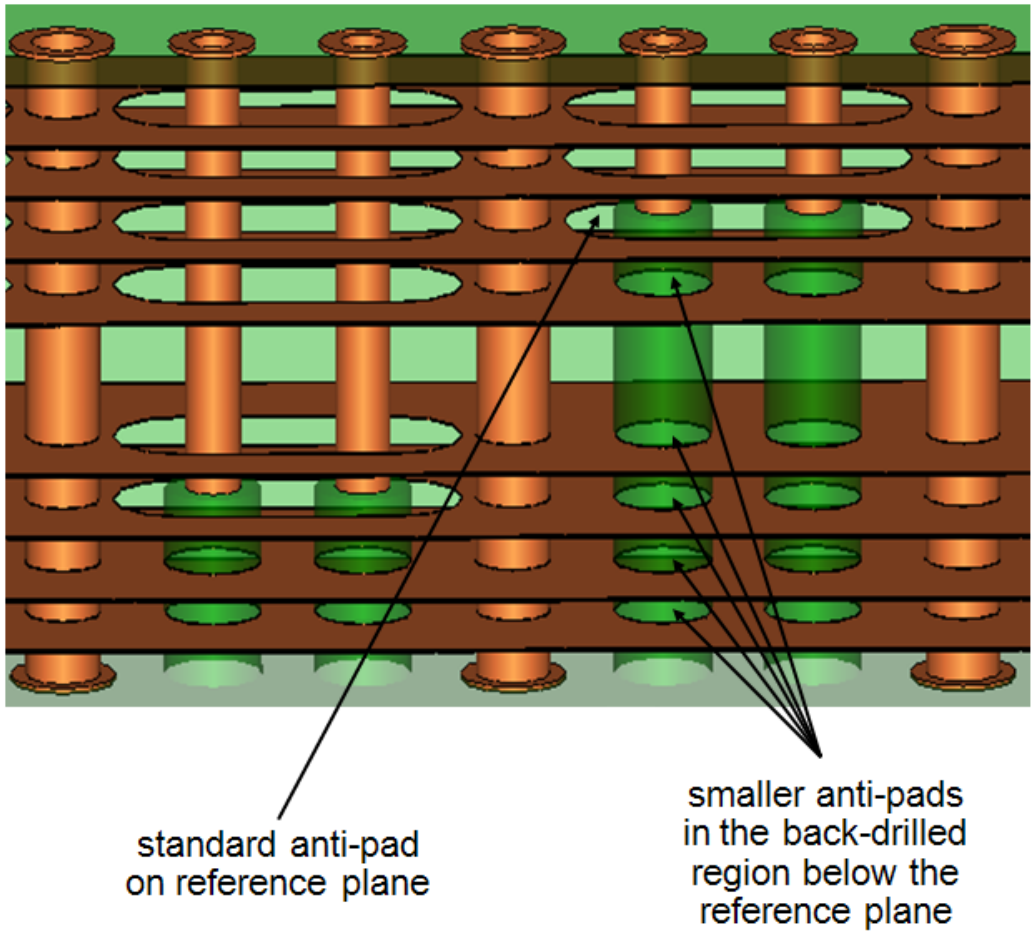


Figure 18: Reduced anti-pad size in back-drilled regions

5.0 APPLICATION PROCEDURE

5.1 Connector Insertion Tooling

The ExaMEZZ connectors are intended to be applied utilizing a “flat rock” with special insert tool listed in the table below. If the connector’s ground tails are longer than the thickness of the daughter card that the connector is being applied to (ground tail length specification is 1.60 ± 0.15mm) a special bottom support tool will be necessary. This tool could be a PCB with oversized holes or a custom tool designed by the user.

Insertion Tool Product Number	Number of Pairs	Number of Columns	Number of Positions
10125494-015	2-Pair	15	120
10125494-019	2 Pair	19	152
10125496-011	4 Pair	11	154
10125496-025	4 Pair	25	350

Table 5: Insertion Tool Product Numbers

5.2 Insertion Presses

Several important items to consider when selecting an insertion press include:

- The press must have sufficient force capabilities.
- The press must be capable of controlling the insertion rate.
- The press must be capable of pressing per a force gradient curve.
- The press ram must be large enough to cover the insertion tooling.
- The press table must be large enough to properly accommodate the PCB size.

FCI recommends using a servo driven electronic press from the Tyco Electronics (ASG) MEP family of presses. See section 5.3 for programming details.

5.3 Recommended Insertion Press Settings

The recommended maximum insertion force is 12 Newton per press-fit terminal. FCI recommends using a 75 degree force gradient press profile to ensure that the connectors are fully seated, while at the same time are not damaged by over pressing the connectors onto the board. A typical 75 degree press profile would look like Figure 19 below. The recommended insertion rate is 0.050”/sec for the entire time the ram is in contact with the connector press tool.

Note: FCI RECOMMENDS THAT ONLY ONE CONNECTOR ASSEMBLY BE INSERTED AT A TIME. If more than one connector is inserted at the same time then extra care must be taken to ensure that the push surface of the tool is parallel to the component surface of the board.

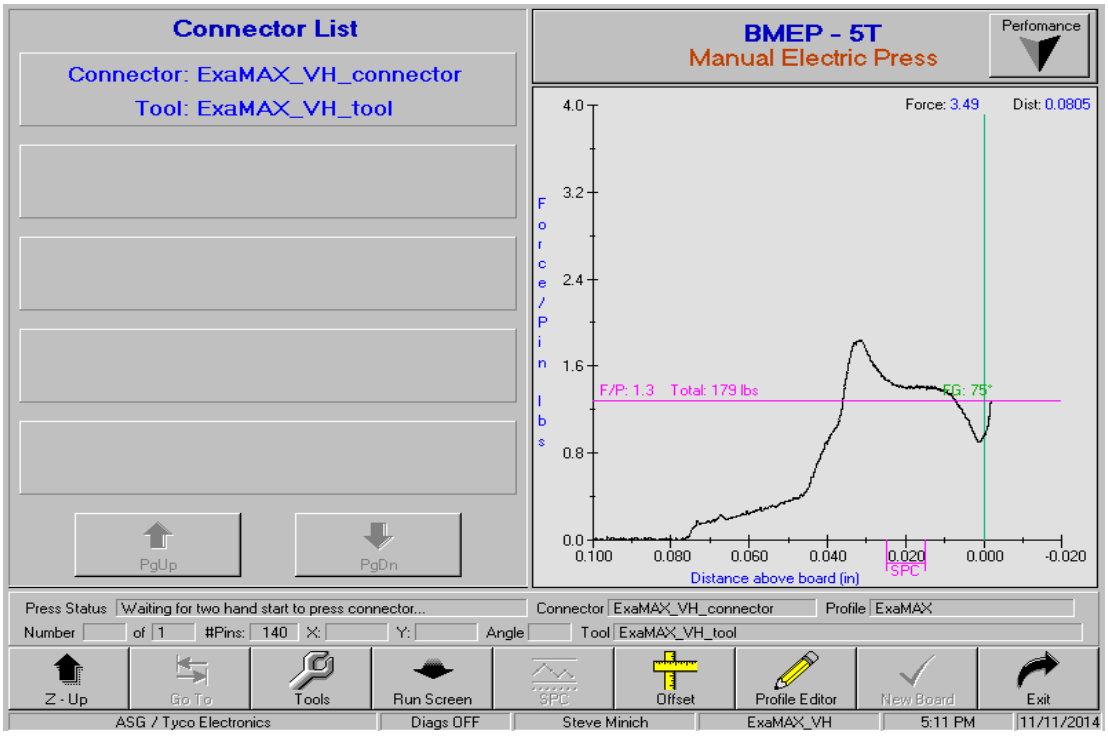


Figure 19: Example of 75 Degree Force Gradient Press Profile

5.4 Board Insertion Procedure for ExaMEZZ connectors

- Place insert tool into the connector assembly and set in the desired location on the printed circuit board taking care to assure that all press-fit tails line up with the proper holes and each cavity slot of the connector has a blade from the insert tool located in it.

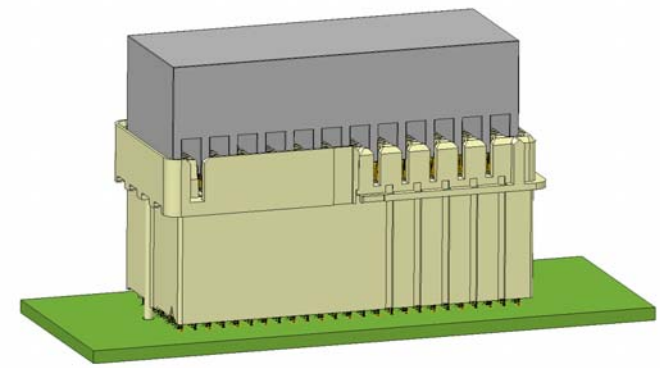


Figure 20: Placing Insert Tool into Connector

- Using a flat rock surface and an insertion press, apply force to the top of the insert tool as shown in the picture below.

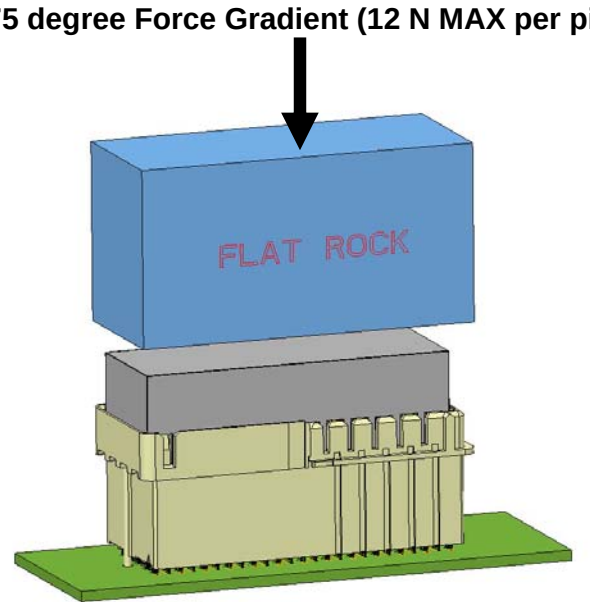


Figure 21: Pressing Connector to PCB

- Inspect for proper application (refer to section 5.5).

5.5 Post-Application Inspection Procedures

The post-application inspection should consist of several simple checks to assure that the connector is applied properly and is not damaged.

- Visually assure that all press-fit tails are seated in the proper PCB holes and that none have been crushed during application.
- Visually check that the standoffs on the bottom of each assembly are seated within 0.10 mm of flush to the PCB (see Figure 22). A larger gap beneath the standoffs or retainer may indicate that the connector is not seated fully or is not seated parallel to the board.

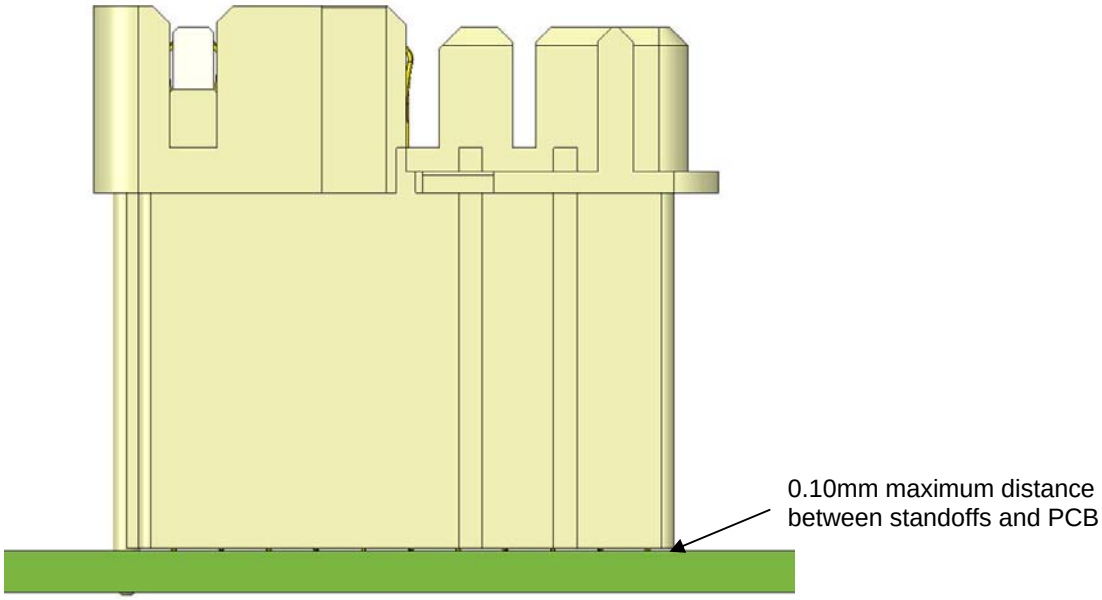


Figure 22: **Proper Seating after Board Application**

5.6 Connector Removal Tooling

The following are FCI removal tool part numbers for ExaMEZZ connectors. Due to size restrictions the removal tool only presses on the ground press-fit tails.

ExaMEZZ Hermaphroditic	Column pitch, mm	Column 1 IMLA	Removal Tool part no.*
2 Pair / 15 Column	2.0	B	10125499-015
2 Pair / 19 Column	2.0	B	10125499-019
4 Pair / 11 Column	2.0	B	10125509-011
4 Pair / 25 Column	2.0	B	10125509-025

Table 6: ExaMEZZ Removal Tools

*** To minimize costs the largest removal tool for a given column pitch may be used to remove any connector with that column pitch by disassembling the tool and removing pins as necessary. Extreme care must be used when doing this since the A1 location is different for ExaMAX® and ExaMEZZ® connectors. Pins only go into the ground holes.**

5.7 Removal Tool Description

The connector removal tool is a hand tool used to remove ExaMEZZ connectors from the back side of the circuit board by pressing on the ground EONs. A hand operated arbor press and bottom support tool (not included with removal tool) are required to actuate the tool and provide support around the connector during the removal process.

It is important to note that the ExaMEZZ Hermaphroditic connector removal tool is red in color (similar to the ExaMAX header). The ExaMAX receptacle tool is blue in color. If the wrong tool is used for rework, damage to the plated through holes and/or the tool may occur.

5.8 Connector Removal Procedure

- A manual arbor press is adequate to perform connector removal
- With the connector to be removed facing downward, place the board assembly over a bottom support tool (not included) that is appropriate for the board layout (see Figure 23).
- It is preferred to support on all 4 sides (2 minimum) of the connector. Also make sure that no small components (chip resistors, chip capacitors, etc.) are between the board and the support tool. The size and shape of the support tooling will vary due to the connector size, number of EONs, whether there are adjacent connectors, and any other surrounding components.
- From the back side of the board, roughly align the A1 side of the tool with the A1 via location on the PCB.
- Final align the tool so that all the pins protruding from the face of the tool are inserted into the ground vias (larger 0.50mm diameter vias) belonging to the connector that you wish to remove. If the pins and vias do not align, rotate the tool 180° and try again. The pins and vias will only align one way. DO NOT TRY TO PUSH PINS INTO SIGNAL VIAS (smaller 0.36mm diameter vias) OR WHERE THERE ARE NO VIAS. THE PINS WILL BUCKEL AND BREAK.

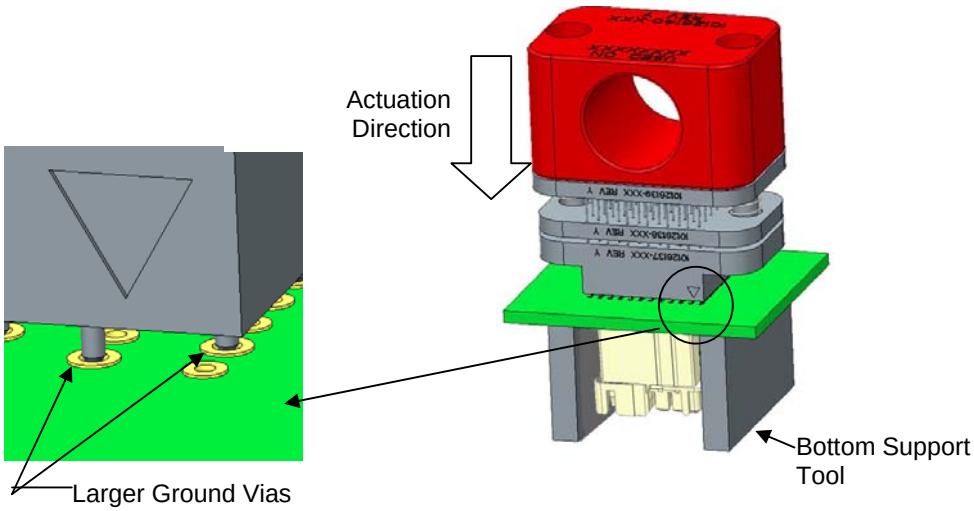


Figure 23: Removal Tool Procedure

- Actuate the press so that the tool is compressed. Apply pressure evenly and slowly, not with a quick motion. Stop applying pressure when the force drops or the press handle becomes easy to move. This will indicate that the EONs have released from the printed circuit board. The connector should be lying on the bench or can be easily removed by hand.

6.0 REFERENCE DOCUMENTS

- FCI PRODUCT SPECIFICATION GS-12-1150
- FCI PRODUCT CUSTOMER DRAWINGS
- FCI PACKAGING SPECIFICATION GS-14-920 (Lead Free Labeling)
- FCI CUSTOMER DRAWING 10119933 (Recommended Via Drill Sizes and Plating)
- ExaMAX ExaMEZZ Power Design Guide GS-20-0456

FCI product drawings and specifications can be obtained by accessing the FCI website www.fci.com or contacting FCI Technical Service. In the event of a conflict between this application specification and the drawing, the drawing will take precedence. Customers are advised to refer to the latest revision level of FCI product drawings for appropriate details.

7.0 NOTES

8.0 RECORD RETENTION

REV	PAGES	DESCRIPTION	EC #	DATE
A		Released		01May14
B	2, 3, 4, 11, 14	Modify sections 3.2, 3.4, 5.1 and 5.6 to add Advanced Mate and new column configurations	ECN-ELX-V-18777-1	01Oct14
C	4,7,8, 10,11, 12	Add Power Section 3.5. Modify sections 5.2-5.4. Max insertion F now 12 N per contact vs 15 N. New section 5.4. Remove Fig 6 Trace routing example 2 and adjust figure numbers.	ECN-ELX-V- 19552-1	25Nov14
D	All	Chg ™ to ®, add section 3.4 PCB Spacers/standoffs and 3.5 Mating/Unmating Methods	ECN-ELX-V- 20050-1	21Jan15
E	9,11	Adjust mating sequence table 4. Correct figure numbers. Adjust spacing and removal tooling clearances in 4.2 and 4.4	ECN-ELX-V- 20737-1	22Apl15
F	11,12	Rename section 4.2, add more descriptive text and add figures for connector spacing and orientation. Move sentence “If there is a need for removal…” to 4.4. Correct figure numbers.	ECN-ELX-V- 21490-1	23Jul15
G	1,9-10, 17-22	Insertion press “must” (vs. should) be capable. Combine 5.4 into 5.3 and reword focusing on 75° Force gradient. Add more bullets to 5.8 (was 5.9) Connector Removal. Add in Power Design Guide. Many others.	ECN-ELX-V- 22269-1	30Oct15