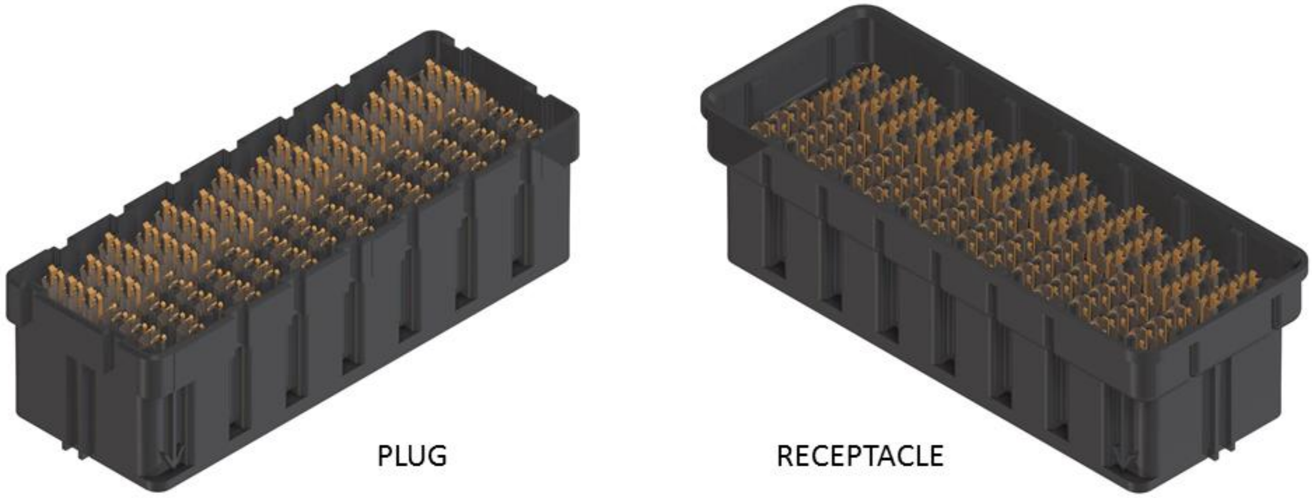
NEOPRESS CONNECTORS



1.0 SCOPE
This specification provides guidance for the use of NeoPress connectors, it is not all inclusive and refinements may be needed that are beyond this document. If questions arise, then please contact Molex.

2.0 PRODUCT DESCRIPTION
NeoPress is a line of Board to Board Connectors that connect one printed circuit board to another printed circuit board to which it's parallel; this type of connector is also known as a "mezzanine" connector. NeoPress is primarily for high speed Differential signals (i.e. Double-Ended signals), but is also appropriate for Single-Ended signals, sundry Low Speed signals and Power connections.

NeoPress can be made in varying heights to accommodate varying separation distances between two parallel boards in a variety of Grids: 2, 3, 4, 6, 8 & 10 Rows **X** 4 - 30 Columns. Check with Molex for availability. Larger configurations available upon request.

NeoPress Plug – Compliant Pin Mountable (Series Number: 172801, 173363 & 203341)
Part Numbers: 172801****, 173363**** & 203341****

NeoPress Receptacle – Compliant Pin Mountable (Series Number: 172832 & 173364)
Part Numbers: 172832****, 173364**** & 203340****

3.0 DOCUMENTS

PRODUCT PRINTS
Plug: 172801-****, 173363-**** & 203341****
Receptacle: 172832-****, 173364-**** & 203340****

SPECIFICATIONS
Product Specification: PS-172801-0001 (100 Ohm) & 2033400001-PS (85 Ohm)
Packaging: PS-173358-0001
Application Tooling Specification Sheet: ATS-622031300 (6x14 Plug Press-in Tool)
Application Tooling Specification Sheet: ATS-622031310 (6x14 Receptacle Press-in Tool)
Application Tooling Specification Sheet: ATS-622031330 (6x10 Plug Press-in Tool)
Application Tooling Specification Sheet: ATS-622031340 (6x10 Receptacle Press-in Tool
Application Tooling Specification Sheet: ATS-622031350 (6x16 Plug Press-in Tool)
Application Tooling Specification Sheet: ATS-622031360 (6x16Receptacle Press-in Tool)
Application Tooling Specification Sheet: ATS-622031370 (8x20 Plug Press-in Tool)
Application Tooling Specification Sheet: ATS-622031380 (8x20 Receptacle Press-in Tool)
Application Tooling Specification Sheet: 2002141180 (4x12 Plug Press-in Tool)
Application Tooling Specification Sheet: 2002141196 (4x12 Receptacle Press-in Tool)
Application Tooling Specification Sheet: ATS-622031410 (6x10 Plug and Receptacle Removal Tool)
Application Tooling Specification Sheet: ATS-622031420 (6x14 Plug and Receptacle Removal Tool)

4.0 APPLICATION TOOL RECOMMENDATIONS

An application tool is required to press-in both the plug and receptacle. Refer to Section 3 above for applicable documents. Other configurations available upon request. See Figure 2 below for an example of correct placement of the press-in tool.

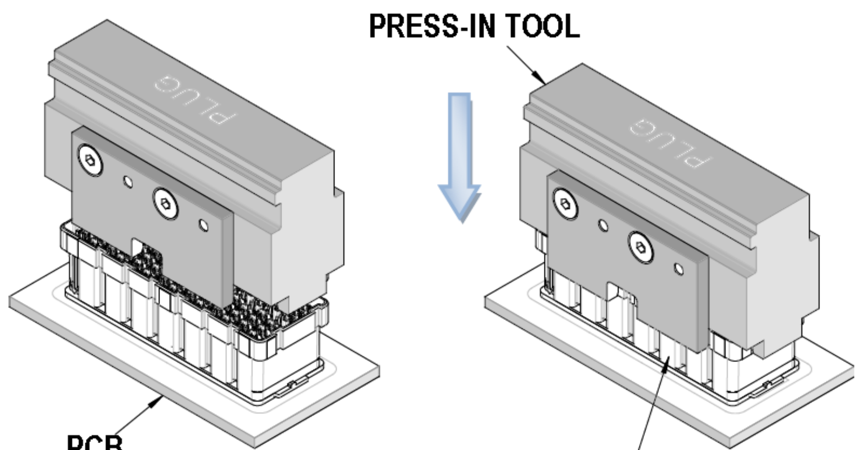


Figure 2

4.1 COMPLIANT PIN PERFORMANCE

Insertion force is 6 lbs. max per compliant pin. This value is intended for press sizing only. Typical peak values are less than 4 lbs. per pin. The peak force value will occur prior to the final seating of the connector. Plating surface finish and PCB materials will impact actual values.

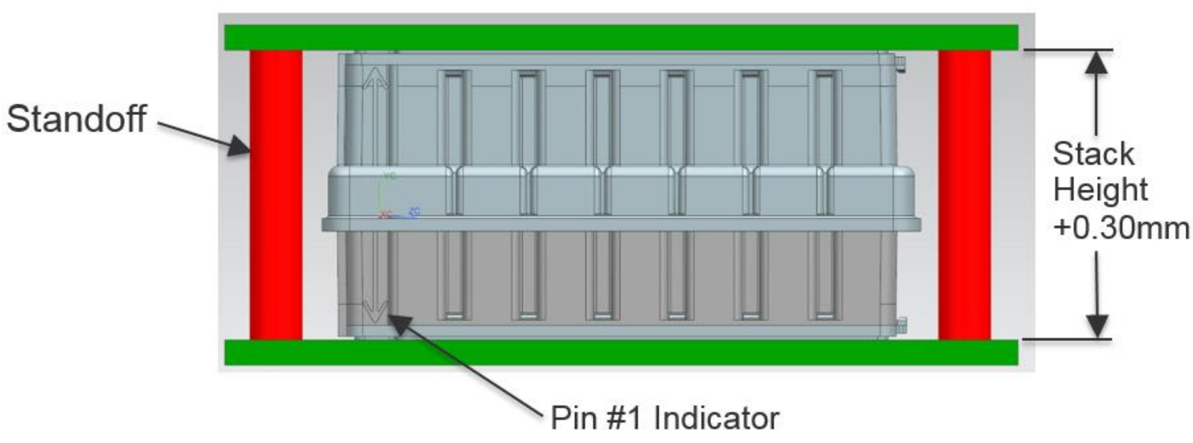
Retention force is .8 lbs. min. per compliant pin. This reflects minimum expected values for retention forces when tested in plated through holes drilled and plated as described in section 6.0. Plating surface finish and PCB materials will impact actual values.

5.0 GENERAL REQUIREMENTS

5.1.1 ASSEMBLY INSTRUCTIONS

NeoPress as with any mezzanine connector of this type will because of its rigidity and tight mechanical fit, determine the lateral position of the daughter board with respect to the mother board. So any assembly fixture used to mate two boards having NeoPress connectors must allow the mating connectors to seek their natural mated condition without imposing undue side forces. For example: If an assembly fixture enforces a prescribed lateral alignment between the two PCB's based upon criteria such as the board edges, then the connectors may be damaged during the mating process.

There is no connector/connector latching in NeoPress connectors, so the two mated printed circuit boards must be held together by other means such as standoff's and screws. But any device such as a standoff, which is used to fix the daughter board to the mother board must allow for tolerances so that undue side forces don't arise. See images below for recommended standoff placement and height. Placement of standoffs should be as close to the connector as possible. Standoff height should be the stack height plus 0.30mm to allow for PCB flatness and connector tolerances.

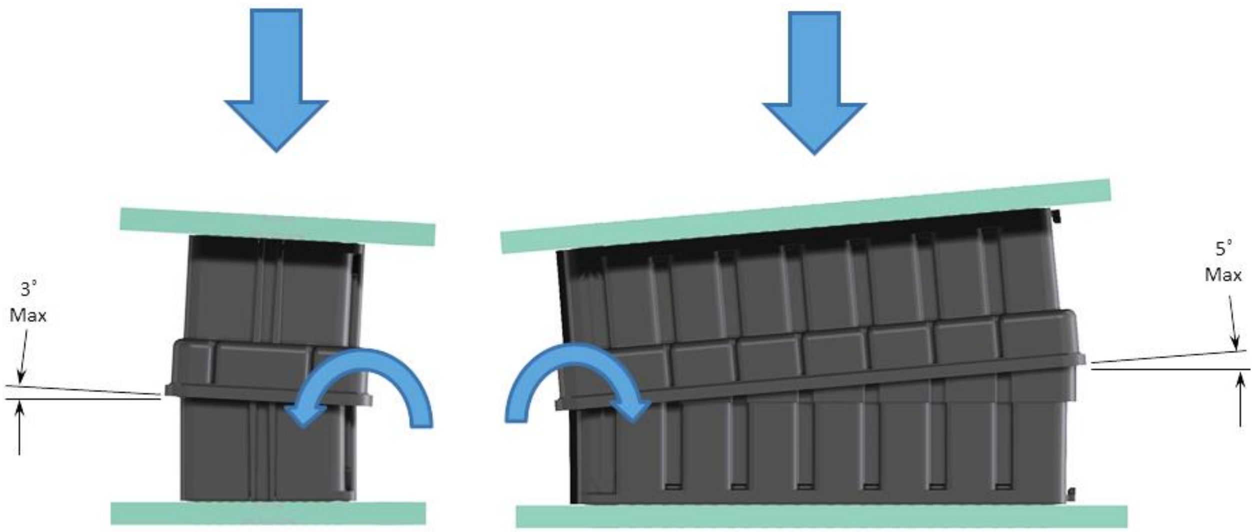


5.1.2 RECOMMENDED ANGULAR ALIGNMENT FOR MATING

To minimize risk during mating:

I) Connectors should be parallel with respect to each other during mating.
II) Use a smooth motion during mating (No mechanical shock).

If necessary to mate at an oblique orientation; then the recommended maximum skew:



If a fixture is used to do the mating, then that fixture should hold the mating connectors parallel to within +/- 2 degrees. Also, the fixture should allow the Connectors to become parallel as the mating process progresses.

5.1.3 RECOMMENDED LATERAL ALIGNMENT FOR MATING

To minimize risk during mating, the assembly process should target a zero off-set from the exact connector alignment. Also, use a smooth motion (No mechanical shock) during mating. The connector “lead-in’s” will correct for linear off-sets as shown below:

MAXIMUM MISALIGNMENT FOR MATING
(Above two images are not to scale)

5.1.4 UN-MATING

Connectors can be un-mated more easily by gently rocking the mated printed circuit boards (back and forth) while simultaneously pulling apart. Recommend that this rocking motion be limited to be within +/- 2 degrees from parallel.

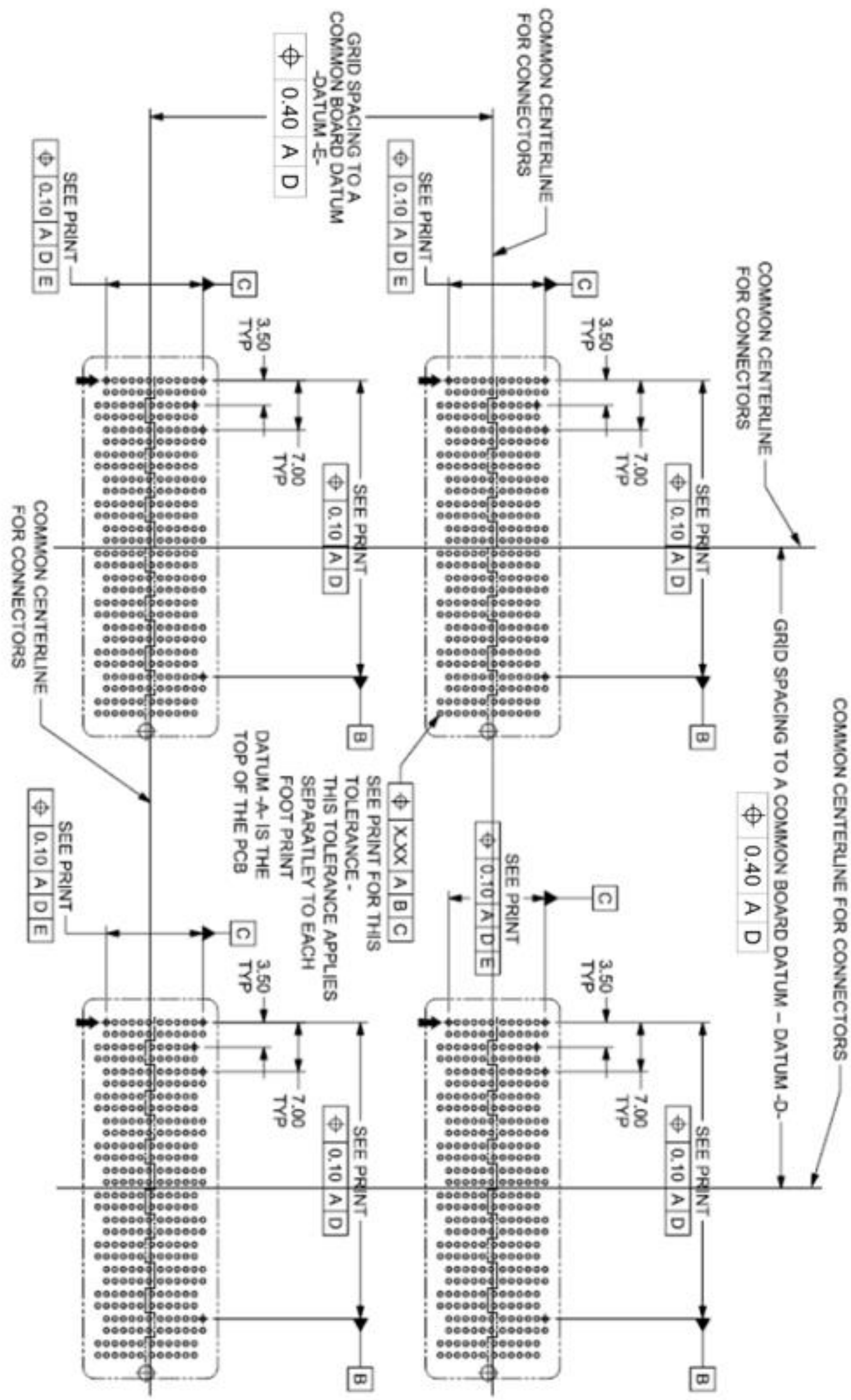
5.1.5 MULTIPLE CONNECTORS

All connectors on a given board must be the same gender and be applied in the same orientation as shown (See the applicable prints for foot print dimensions). Also, the connectors must come from the same package (Or successive packages), and have the same manufacturing date.

The below illustration employs four connectors, but this pattern may be extended to incorporate additional connectors. Regardless of the quantity of connectors on these PCB’s, the risk of differing thermal coefficients of expansion between the Mother and Daughter Boards must be assessed to avoid overstressing the Compliant Pins.

APPLICATION SPECIFICATION

5.1.5 MULTIPLE CONNECTORS (CONTINUED)

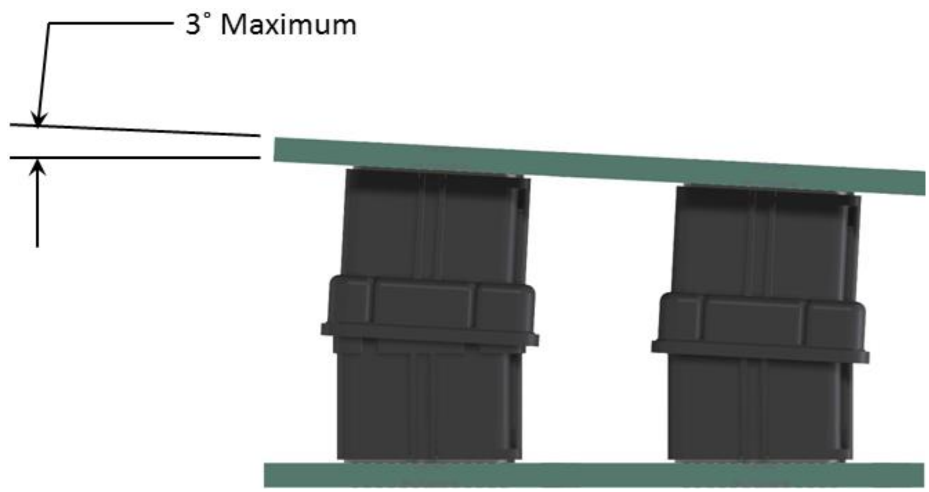


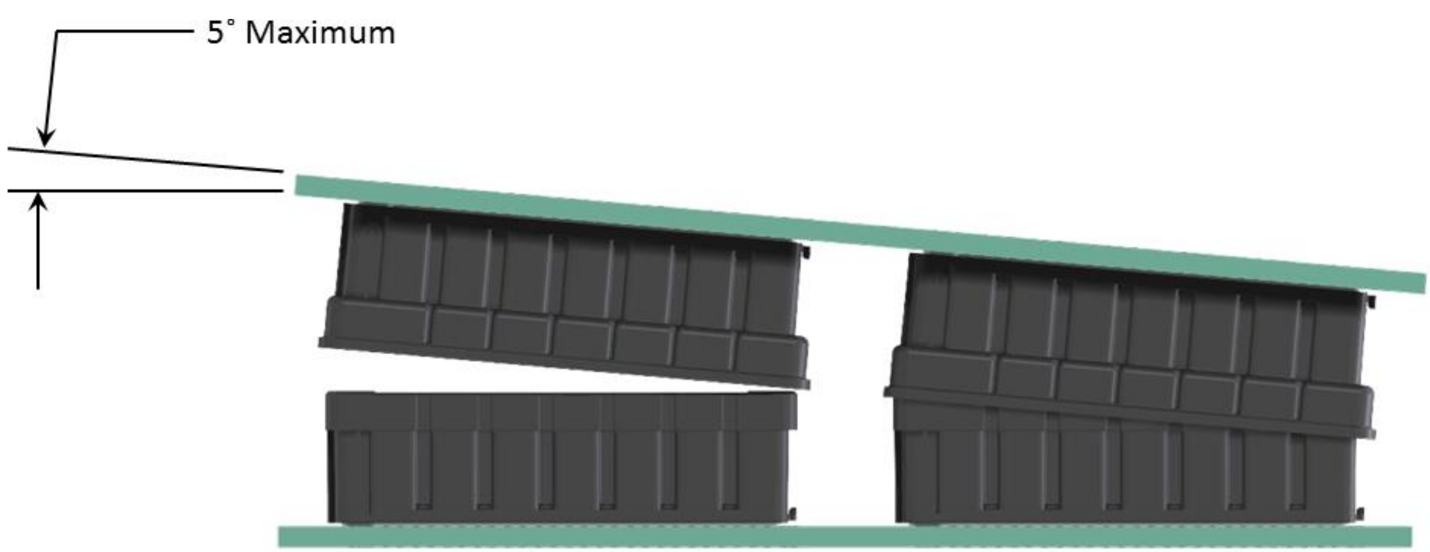
REVISION:	ECR/ECN INFORMATION:	TITLE:	SHEET No.
D	EC No: 177816 DATE: 2018/06/07	NEOPRESS MEZZANINE CONNECTORS	6 of 19

DOCUMENT NUMBER:	CREATED / REVISED BY:	CHECKED BY:	APPROVED BY:
AS-172801-0001	PREMO	PELOZA	LANG

5.1.5 MULTIPLE CONNECTORS (CONTINUED)

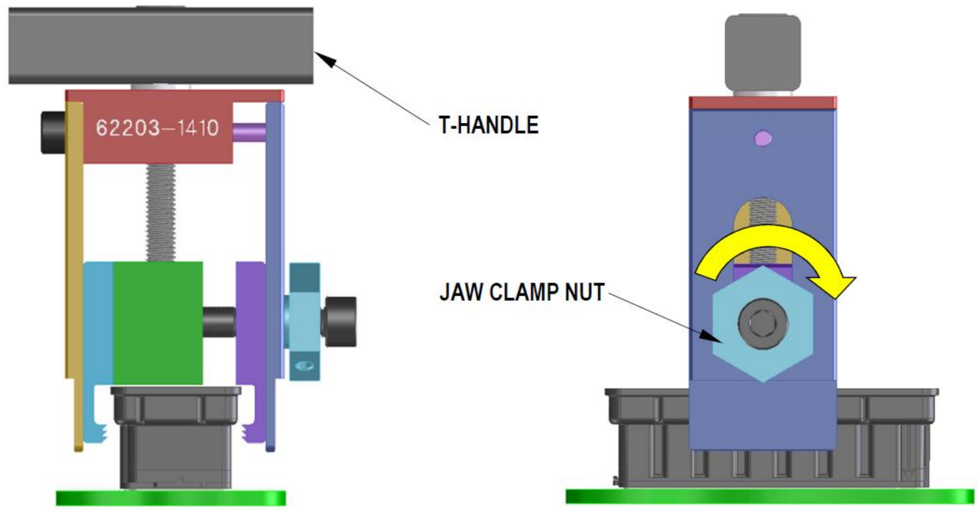
The recommendation for oblique mating for multiple connectors is similar to that for a single mated connector set:





5.1.6 REWORK CONSIDERATIONS

Connectors may be removed from the PCB with Molex Removal Tool 62203-1410 so long as doing so does not cause enough mechanical strain to damage other components. Refer to section 3 and Molex Application Tooling Specification ATS-622031410 and ATS-622031420. Connectors may be repaired a maximum of 3 times. After the damaged connector is removed, it must be discarded; it cannot be reused.



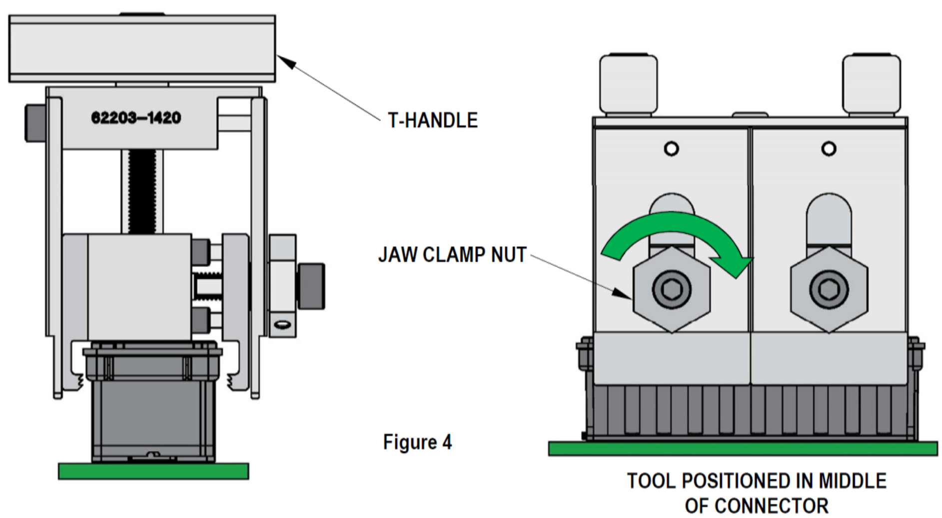
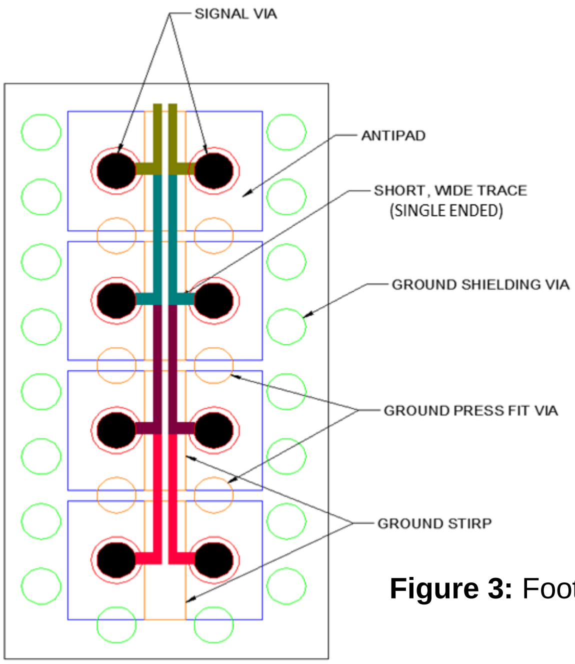


Figure 4

6.1 Recommended Via drill Constraints for NeoPress footprint

1. The NeoPress Triads are arranged as grid. Refer to applicable sales drawings for more information.
2. The connector press-fit via dimension is 0.36mm nominal diameter, with 0.45mm typical drill size. See page 9 for more details.
3. Via optimization is key to improved electrical performance, a through via may be back-drilled to achieve this. Refer to Section 6.2 for back-drill information.
4. A 0.30mm via stub length is recommended, 0.46mm maximum via stub length is allowed for 28Gbps.
5. Tune the additional ground vias to the optimum diameter. Typically, minimum drill diameter is best for 100-ohm impedance and larger drill diameter is best for 85-ohm impedance. Ground shielding vias make board crosstalk small. Refer to Figure 3 for a typical footprint pattern with additional ground vias.
6. 1.65mm signal-to-signal via distance
7. 1.2mm signal-to-ground via distance
8. 1.2mm stagger to reduce crosstalk



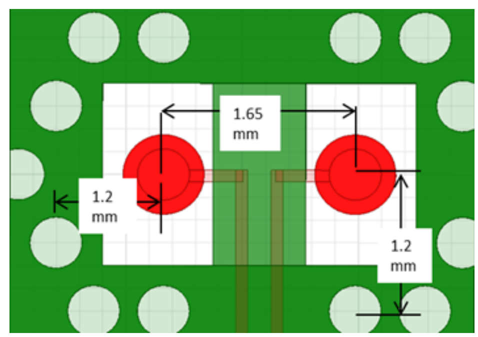
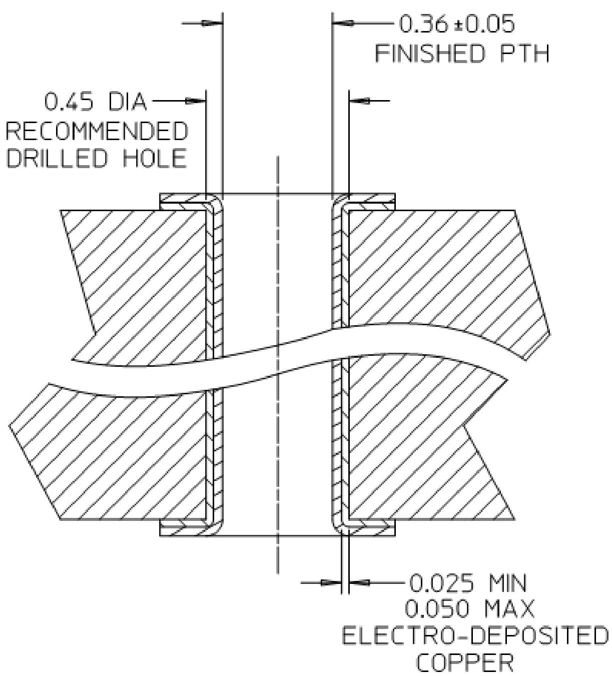


Figure 3: Footprint Pattern

The recommended pad stack for the hole size is contained in Table 1. All non-functional pads are to be removed for high speed applications.

FEATURE	0.36mm PTH NOMINAL DIA
Finished hole	0.36mm (14.2 mil)
Recommended Drill	0.45mm (17.7 mil)
Interior Pad	0.70mm (27.6 mil)
Top Layer Pad	0.70mm (27.6 mil)
Bottom Layer Pad	0.70mm (27.6 mil)

Table 1
Pad Stack Dimensions



Notes:

1. The finished PCB hole size is the critical feature for proper performance of the compliant pin terminal. The drill sizes listed are recommended based on Molex's qualification to achieve the finished PCB hole size.
2. Depending upon the specific manufacturer's plating process, a larger drill size may be used to better target the nominal finished PCB hole size.
3. The typical drill hole tolerance is +/-0.013mm.

6.2 Back-drill

The size of back-drill hole should be large enough to remove all the copper of the annular ring that surrounds the via being modified. Molex typically doesn't recommend the exact size, but usually this drill diameter is 0.10mm (0.004") larger than the outside diameter of the annular ring. Figure 4 shows the required minimum via length from top of PCB to be remained after back-drill.

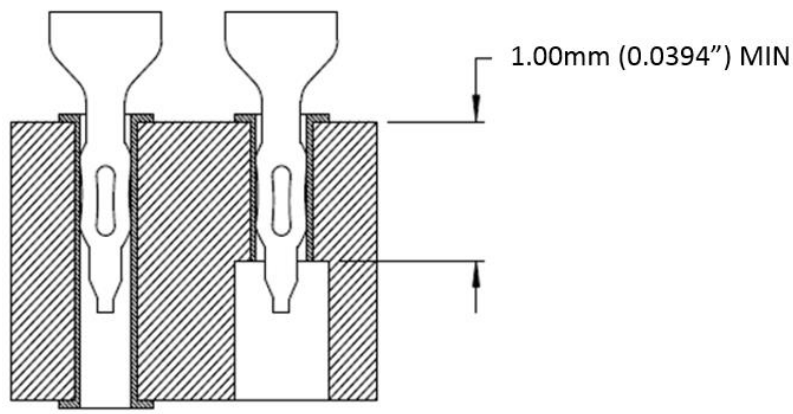


Figure 4: Minimum via length after back-drill

6.3 Example of Connector Pin configuration pattern

		1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
	A	28G	28G			28G	28G			28G	28G			28G	28G		
	B	GND	GND			GND	GND			GND	GND			GND	GND		
	A			28G	28G			28G	28G			28G	28G			28G	28G
	B			GND	GND			GND	GND			GND	GND			GND	GND
	C	SE	SE			28G	28G			28G	28G			28G	28G		
	D	GND	GND			GND	GND			GND	GND			GND	GND		
	C			28G	28G			28G	28G			28G	28G			SE	SE
	D			GND	GND			GND	GND			GND	GND			GND	GND
	E	SE	SE			SE	SE			SE	SE			SE	SE		
	F	GND	GND			GND	GND			GND	GND			GND	GND		
	E			CLK	CLK			POWER	POWER			POWER	POWER			POWER	POWER
	F			GND	GND			POWER	POWER			POWER	POWER			POWER	POWER
MIRROR LINE	G	GND	GND			POWER	POWER			POWER	POWER			POWER	POWER		
	H	CLK	CLK			POWER	POWER			POWER	POWER			POWER	POWER		
	G			GND	GND			GND	GND			GND	GND			GND	GND
	H			SE	SE			SE	SE			SE	SE			SE	SE
	J	GND	GND			GND	GND			GND	GND			GND	GND		
	K	DC	DC			28G	28G			28G	28G			28G	28G		
	J			GND	GND			GND	GND			GND	GND			GND	GND
	K			28G	28G			28G	28G			28G	28G			DC	DC
	L	GND	GND			GND	GND			GND	GND			GND	GND		
	M	28G	28G			28G	28G			28G	28G			28G	28G		
	L			GND	GND			GND	GND			GND	GND			GND	GND
	M			28G	28G			28G	28G			28G	28G			28G	28G

Red	28Gbps
Yellow	SE Lower Frequency
Grey	DC Signal
Green	Ground(GND)
Blue	Power(P)
Purple	Clock(CLK)

Figure 5: Example pinout Pattern

6.4 Recommended Anti-pad dimensions for NeoPress footprint:

1. The anti-pad is rectangular in shape and one type of anti-pad can be used for all layers.
2. The size of the rectangle and the distance of the additional ground vias are tuned to get 85 or 100 Ohm impedance.
3. Recommended anti-pad dimensions are shown in Table 2.

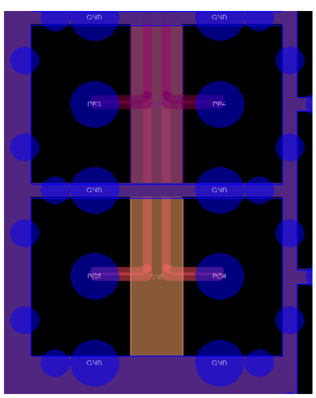


Figure 6: Anti-pad with trace and Ground strip

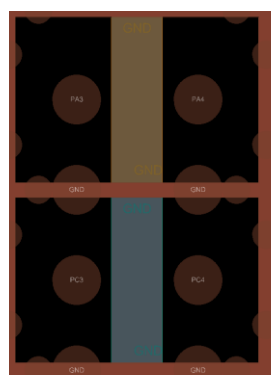


Figure 7: Anti-pad with only Ground Plane

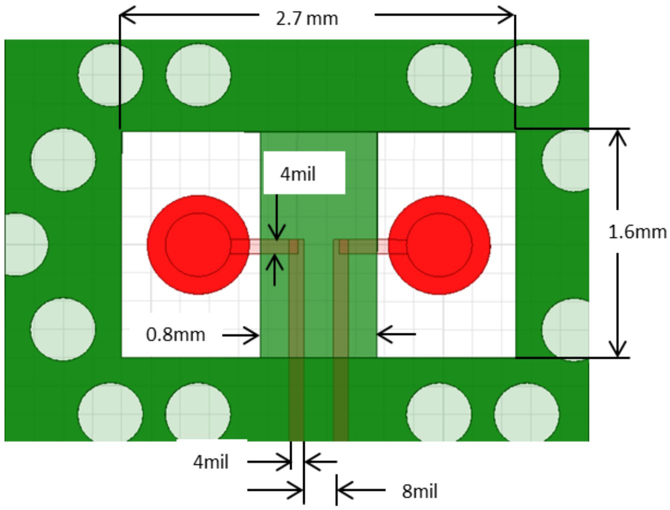


Figure 8: Anti-pad for 100-ohm

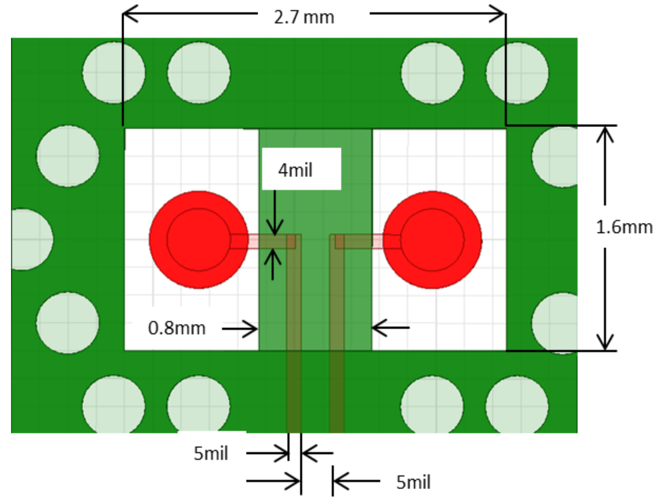


Figure 9: Anti-pad for 85-ohm

Feature	85-ohms	100-ohms
Gnd via diameter	0.50mm	0.25mm
Antipad_h	1.6mm	1.6mm
Antipad_w	2.7mm	2.7mm
Short SE trace	4 mils	4 mils
Diff trace w	5 mils	4 mils
Diff trace space	5 mils	8 mils
Ground strip w	0.8mm	0.8mm
*Note: These dimensions are for reference purpose. Perform optimization as per the stack-up used.		

Table 2: Summary for recommended dimensions

6.5 Recommended routing for high speed differential signal trace.

1. Use symmetric signal traces.
2. Use zero skew traces.
3. Signal trace width is increased to tune impedance within the anti-pad. On signal reference layers, a ground strip is used for impedance control and good ground return.
4. Routing can be done with through vias, or with back drilled vias.
5. Stair-step signal layers with back-drill can provide improved electrical performance.
6. Short section of single-ended trace from via break-out may need to be tuned for impedance control.
7. Trace bending angle, α ≥ 45degrees. Refer to Figure 10.
   i. Spacing between the same pair, A ≥ 5 x of distance to reference plane.
   ii. Length segment B, C ≥ 5 x Trace Width.

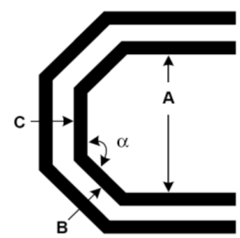


Figure 10: Trace bending angle

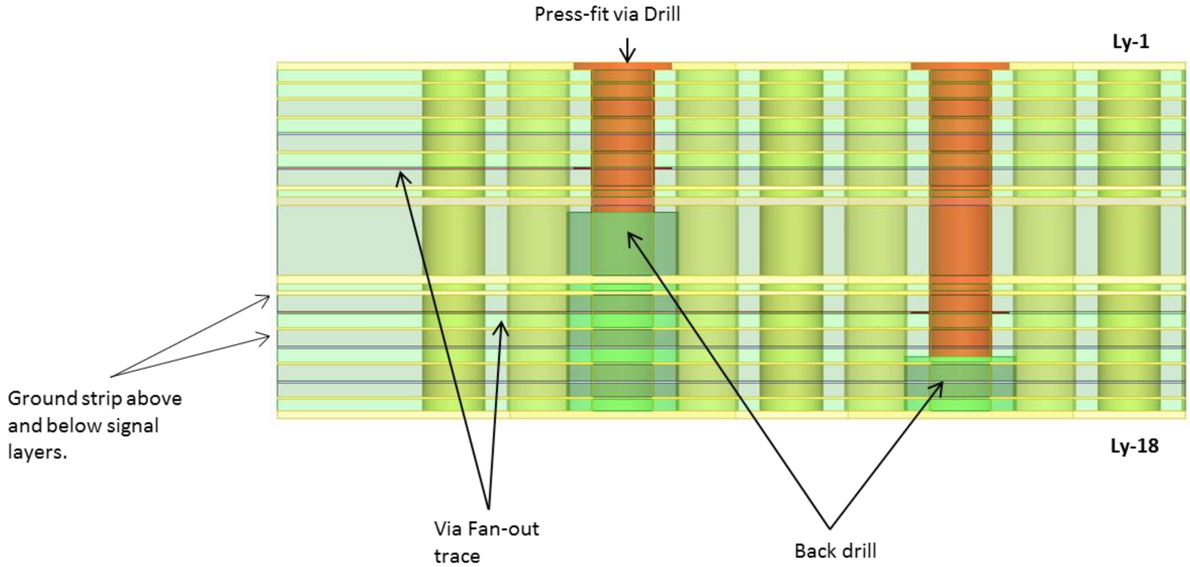


Figure 11: Stair-step signal layers with back drill

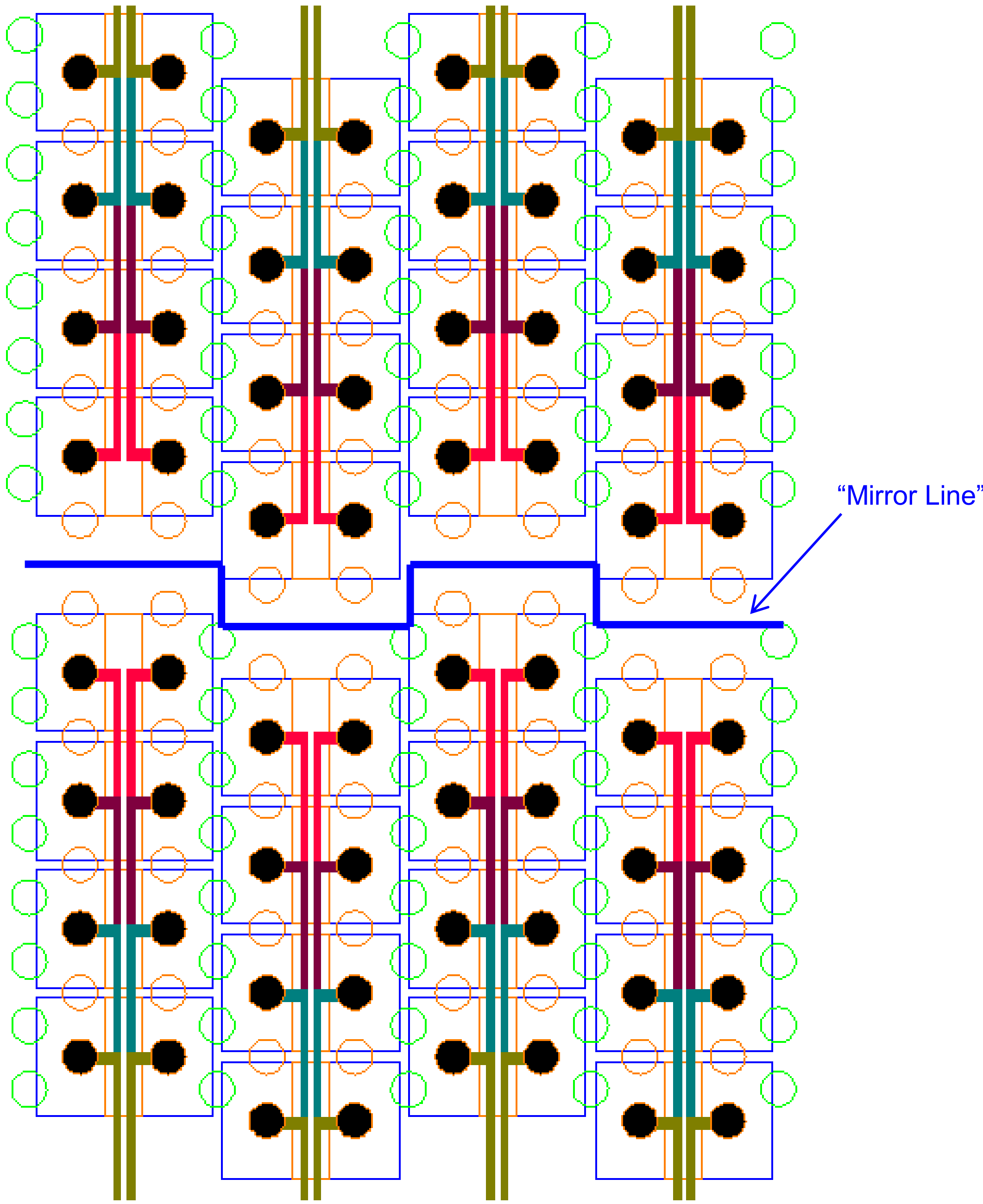


Figure 12: Typical trace escape route pattern

6.6 Example of signal integrity performance plots for via crosstalk and TDR response.

These plots are for board-only simulation results. The simulation uses 18-layer stack-up with D_k = 3.35, D_f = 0.005 and signals on layer 7,12,14,16. The signal vias in the footprint are having back-drill. The 85 and 100 ohm tuning uses the dimensions in Table 2 from section 6.4.

Crosstalk port mapping table

P1 end				P2end		
Diff port #	Footprint Layers	SE Port #	Pin name	SE Port #	Trace fanout Layers	Diff port #
1	Top	1	A1	31	Ly-7	16
		2	A2	32		
2	Top	3	C1	33	Ly-12	17
		4	C2	34		
3	Top	5	E1	35	Ly-14	18
		6	E2	36		
4	Top	7	G1	37	Ly-16	19
		8	G2	38		
5	Top	9	J1	39	Ly-16	20
		10	J2	40		
6	Top	11	A3	41	Ly-7	21
		12	A4	42		
7	Top	13	C3	43	Ly-12	22
		14	C4	44		
8	Top	15	E3	45	Ly-14	23
		16	E4	46		
9	Top	17	G3	47	Ly-16	24
		18	G4	48		
10	Top	19	J3	49	Ly-16	25
		20	J4	50		
11	Top	21	A5	51	Ly-7	26
		22	A6	52		
12	Top	23	C5	53	Ly-12	27
		24	C6	54		
13	Top	25	E5	55	Ly-14	28
		26	E6	56		
14	Top	27	G5	57	Ly-16	29
		28	G6	58		
15	Top	29	J5	59	Ly-16	30
		30	J6	60		

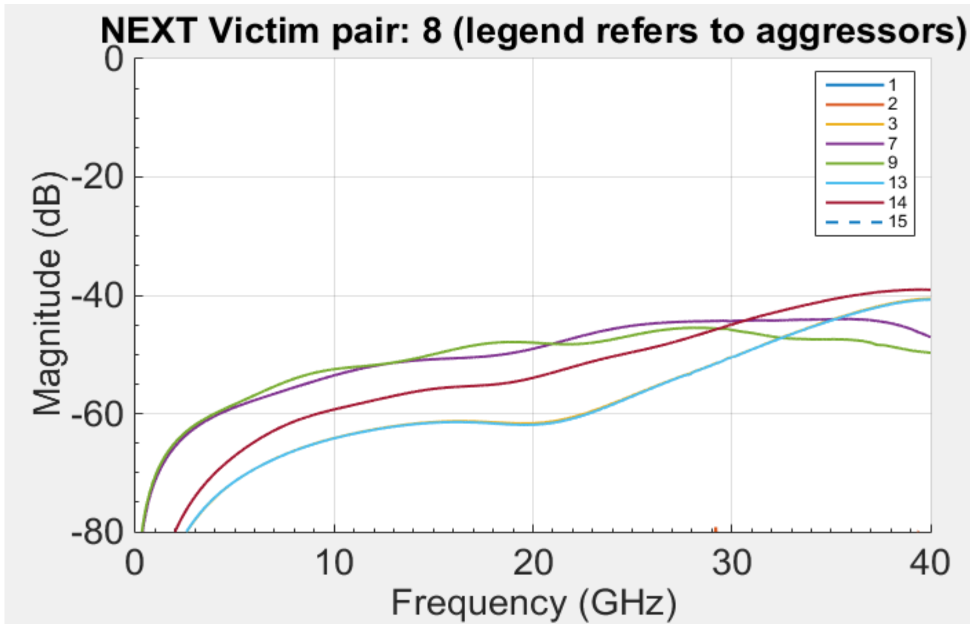
P1 FEXT Mapping Table		
Column 1	Column 2	Column 3
A1	GND	A5
A2	GND	A6
GND	A3	GND
GND	A4	GND
C1	GND	C5
C2	GND	C6
GND	C3	GND
GND	C4	GND
E1	GND	E5
E2	GND	E6
GND	E3	GND
GND	E4	GND
G1	GND	G5
G2	GND	G6
GND	G3	GND
GND	G4	GND
J1	GND	J5
J2	GND	J6
GND	J3	GND
GND	J4	GND

P2 FEXT Mapping Table		
Column 1	Column 2	Column 3
A1	GND	A5
A2	GND	A6
GND	A3	GND
GND	A4	GND
C1	GND	C5
C2	GND	C6
GND	C3	GND
GND	C4	GND
E1	GND	E5
E2	GND	E6
GND	E3	GND
GND	E4	GND
G1	GND	G5
G2	GND	G6
GND	G3	GND
GND	G4	GND
J1	GND	J5
J2	GND	J6
GND	J3	GND
GND	J4	GND

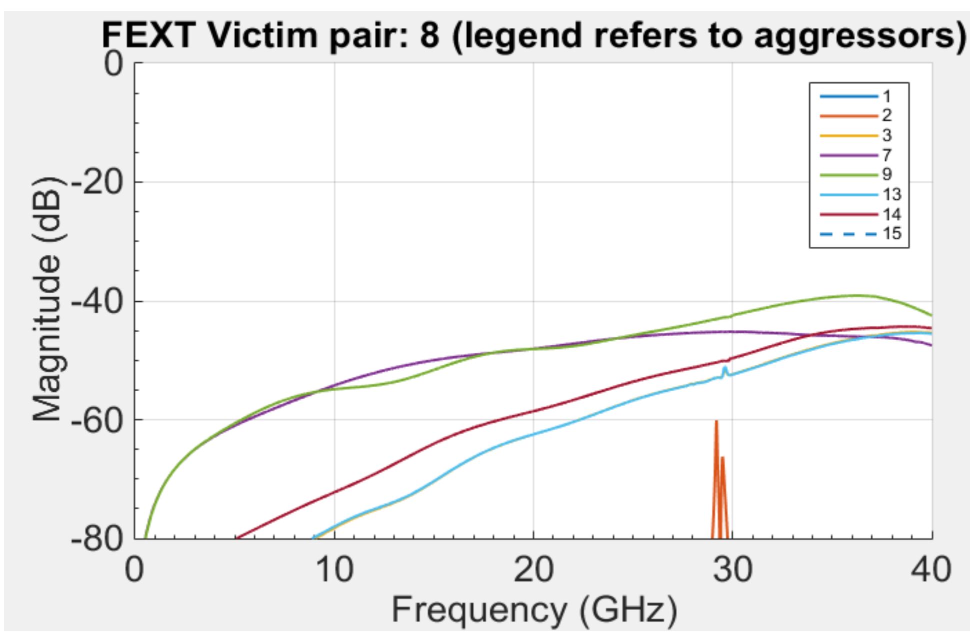
P2 NEXT Mapping Table		
Column 1	Column 2	Column 3
A1	GND	A5
A2	GND	A6
GND	A3	GND
GND	A4	GND
C1	GND	C5
C2	GND	C6
GND	C3	GND
GND	C4	GND
E1	GND	E5
E2	GND	E6
GND	E3	GND
GND	E4	GND
G1	GND	G5
G2	GND	G6
GND	G3	GND
GND	G4	GND
J1	GND	J5
J2	GND	J6
GND	J3	GND
GND	J4	GND

Frequency Domain Plots

Differential NEXT (100ohms)

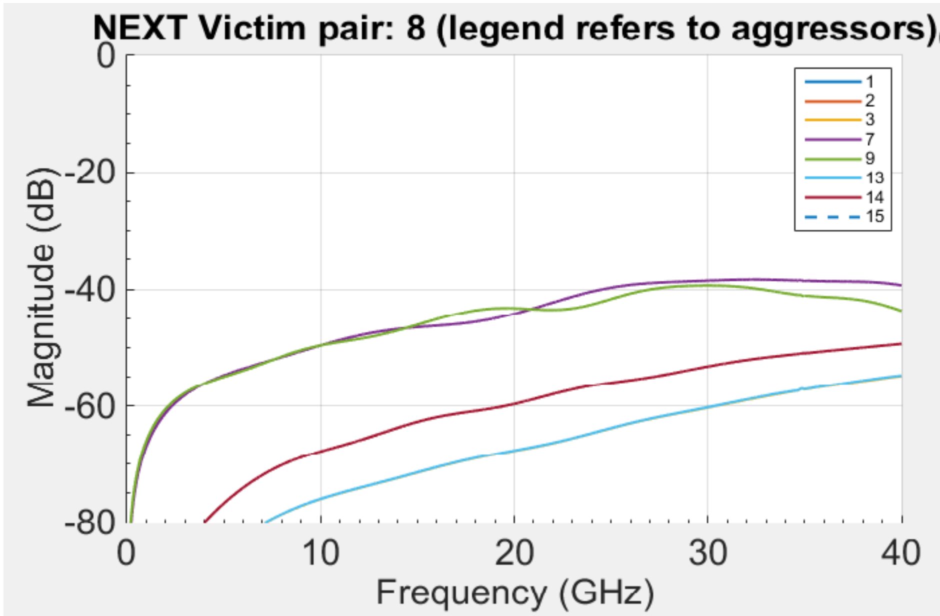


Differential FEXT (100ohms)

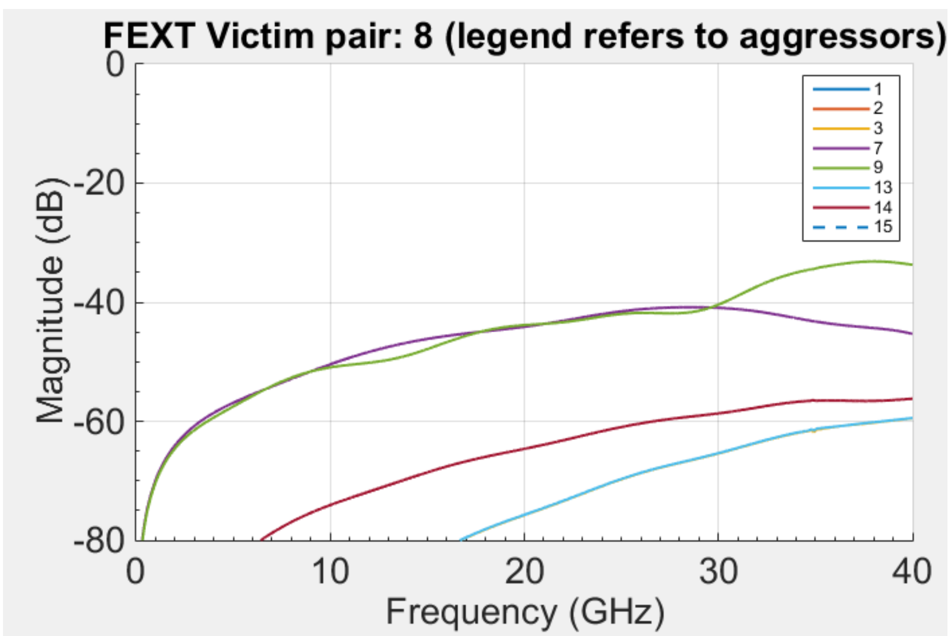


Frequency Domain Plots (Continued)

Differential NEXT (85ohms)



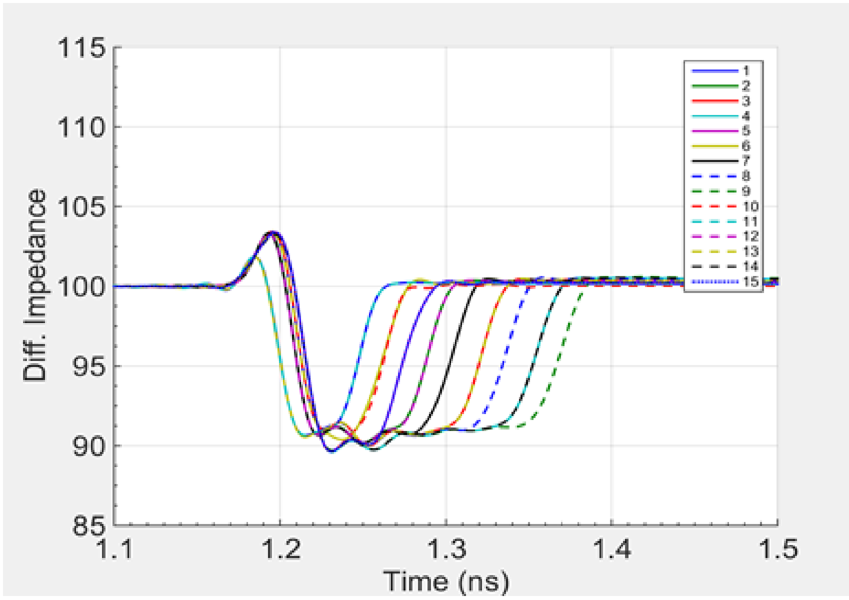
Differential FEXT (85ohms)



Time Domain Plots

Differential TDR Response (100ohms)

- **20ps (20%-80%) rise-time**



Differential TDR Response (85ohms)

- **20ps (20%-80%) rise-time**