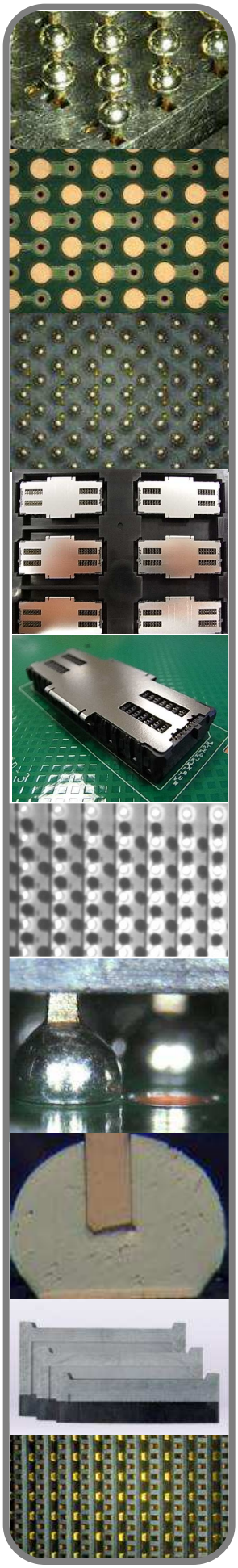
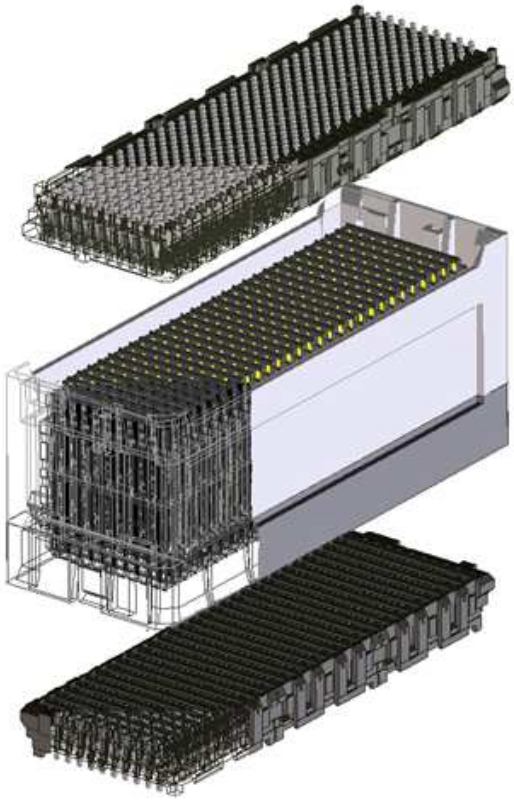


Hirose *IT3/IT5* ™

Connector System



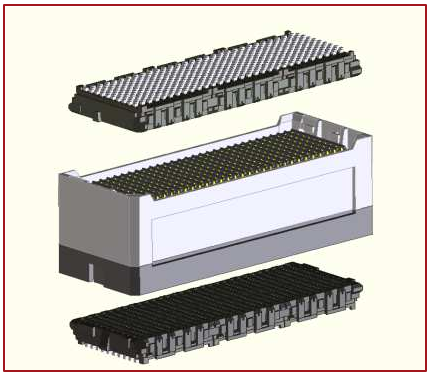
Winner of 2009 SMT Vision Award

GENERAL INFORMATION - DESIGN NOTES - PWB ASSEMBLY NOTES - TIPS FOR SMT ASSEMBLY - SYSTEM ASSEMBLY - SYSTEM DISASSEMBLY

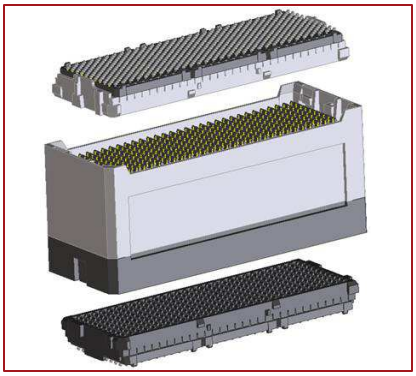
Revision No.	***Description (Major changes)***	***Date***
1.0	Initial release	April 6, 2009
1.1	Revised spacer and nut tightening information	May 18, 2009
1.2	Revised information on 300 position receptacles to include metal stiffening cap	August 26, 2009
1.5	Revised stack height charts, press force information and other information with respect to design note release	November 30, 2009
1.55	Updated the front page and revised information on footprint	December 30, 2009
1.56	Updated rework and packaging information	December 13, 2010
1.6	Updated height variation chart and interposer installation cap	February 24, 2011
2.0	Included information on plugs and an interposer removal tool	May 1, 2011
3.0	Integrated IT5 information	July 15, 2011
3.10	Included IT5H information	May 21, 2012

Section 1 Introduction

*The Hirose **IT3/IT5** connector system is a three-piece mezzanine connector. Process-friendly BGA receptacles are assembled onto PWBs, and separate, configurable interposers complete the connections between circuit boards. It is available in 100, 200 and 300 signal model. IT3 is available with tin-lead or lead-free alloy solder balls and IT5 is available in lead-free alloy solder balls.*



Hirose *IT3* connector assembly



Hirose *IT5* connector assembly

1.1 Purpose

This technical bulletin is intended to provide basic information and product features of the Hirose ***IT3/IT5*** BGA connector system. By providing this information, Hirose believes it can help its customers to speed product development, improve quality and reliability, and limit overall system costs.

1.2 Scope

This guideline provides information useful for applications using the ***IT3/IT5*** BGA connector system. It provides information pertaining to:

a) General overview of the connector system
b) Configuration and part numbering
c) PWB design
d) Assembly processing, inspection and repair

This document will be updated by Hirose as required to reflect current technologies and manufacturing capabilities.

1.3 Application and Interpretation

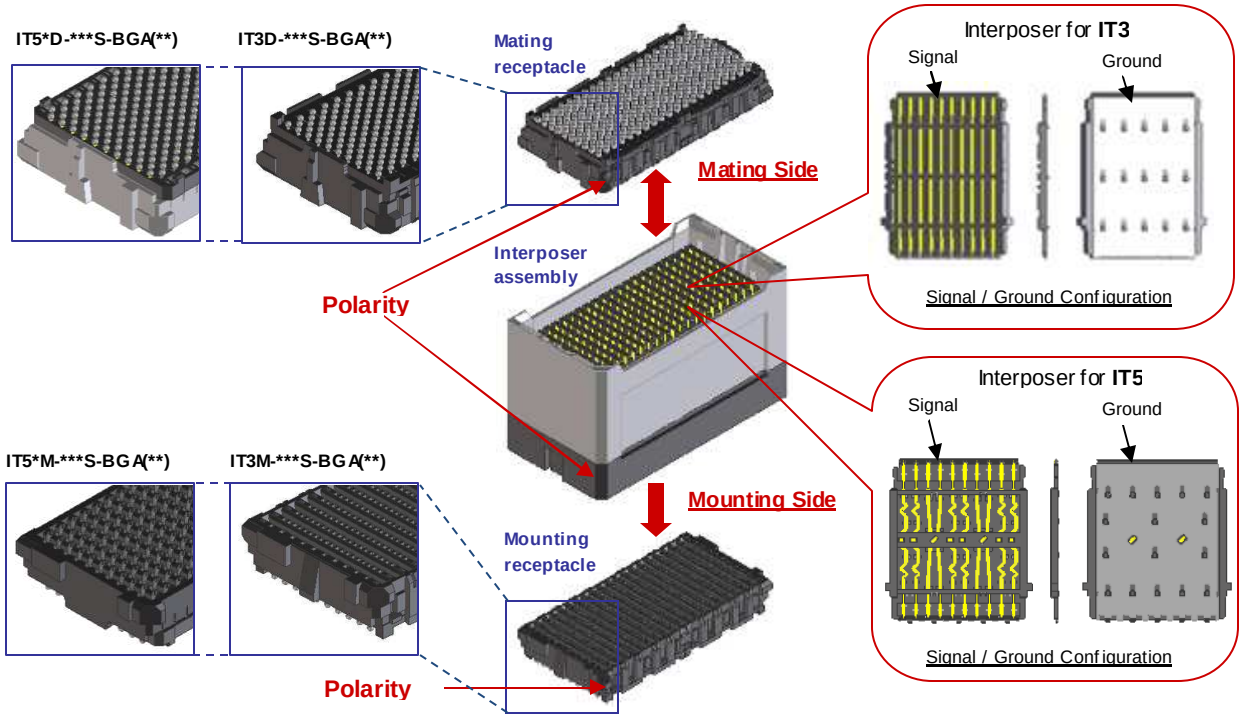
This technical bulletin is intended to offer only general guidance and design concepts to customers. Therefore, it does not limit customer designs nor guarantee results under all situations. Development of actual designs is the responsibility of each customer. Customers should consult with Hirose regarding their specification, when, or if, any questions arise relating to these guidelines. Use of this technical bulletin is at customer's sole risk. This bulletin is provided *"AS IS"* and without warranty of any kind and Hirose *EXPRESSLY DISCLAIMS ALL WARRANTIES, EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. HIROSE DOES NOT WARRANT THAT THE GUIDELINES CONTAINED IN THIS BULLETIN WILL MEET ANY CUSTOMER'S REQUIREMENTS. FURTHERMORE, HIROSE DOES NOT WARRANT OR MAKE ANY REPRESENTATIONS REGARDING THE USE OR THE RESULTS OF THE USE OF INFORMATION CONTAINED IN THIS BULLETIN IN TERMS OF CORRECTNESS, ACCURACY, RELIABILITY, OR OTHERWISE. UNDER NO CIRCUMSTANCE SHALL HIROSE OR ITS DIRECTORS, OFFICERS, EMPLOYEES OR AGENTS BE LIABLE FOR ANY INCIDENTAL, SPECIAL OR CONSEQUENTIAL DAMAGES (INCLUDING DAMAGES FOR LOSS OF BUSINESS, LOSS OF PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION AND THE LIKE) ARISING OUT OF THE USE OF THE INFORMATION CONTAINED IN THIS BULLETIN.*

Section 2 General Information

*Hirose's **IT3/IT5** connector system is designed to provide modular high-speed differential, single-ended and power connections between two parallel boards. The interconnection to the PWBs utilizes process-friendly Ball Grid Array receptacles, while the stacking height of 15 to 40mm for IT3 and 18 to 40mm for IT5 is set by an impedance-controlled interposer that is added at the system level.*

*The **IT3/IT5** connector system consists of two receptacles and one interposer. The receptacles have low profiles and open bodies. The BGA balls are mounted on compliant pins and are set on a staggered grid of 1.5 and 1.75 mm pitch. Both the mating and mounting receptacles' footprints are compatible with popular mezzanine connectors. Receptacles are available in tin-lead and lead-free configurations for IT3 and lead-free for IT5. They can be used in no-clean or water-wash assembly processes.*

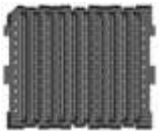
The interposer is an assembly consisting of individual wafers, each carrying 10 signal and 9 ground connections for IT3, and 10 signal and 11 ground connections for IT5. The interposer is mounted to the receptacles and locked in with mechanical latches to create highly reliable and stable mechanical and electrical connections. IT3 and IT5 interposers and receptacles are interchangeable.



2.1 Component Weights

IT3 Mating Receptacle

IT3 D-100S-BGA



IT3 D-300S-BGA

** Receptacle will accept any available stacking height*

Contact Positions	Part Number	Weight
100 (100 signals/90 grounds)	IT3D-100S-BGA(**)	2.5 g
200 (200 signals/180 grounds)	IT3D-200S-BGA(**)	4.7 g
300 (300 signals/270 grounds)	IT3D-300S-BGA(**)	9.0 g

IT3 Mounting Receptacle

IT3 M-100S-BGA

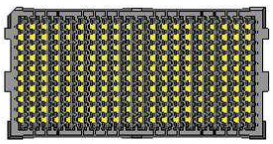
IT3 M-300S-BGA

** Receptacle will accept any available stacking height*

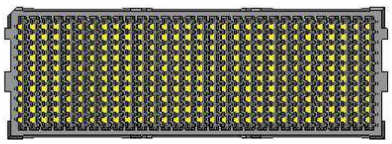
Contact Positions	Part Number	Weight
100 (100 signals/90 grounds)	IT3M-100S-BGA(**)	2.5 g
200 (200 signals/180 grounds)	IT3M-200S-BGA(**)	4.7 g
300 (300 signals/270 grounds)	IT3M-300S-BGA(**)	9.1 g

IT5 Mating Receptacle

IT5 D-200S-BGA



IT5 D-300S-BGA



Contact Positions	Part Number	Weight
100 (100 signals/110 grounds)	IT5D-100S-BGA(**)	2.7 g
200 (200 signals/220 grounds)	IT5D-200S-BGA(**)	5.2 g
300 (300 signals/330 grounds)	IT5D-300S-BGA(**)	9.9 g

** Receptacle will accept any available stacking height*

IT5 +1mm Mating Receptacle

Contact Positions	Part Number	Weight
100 (100 signals/110 grounds)	IT5HD-100S-BGA(**)	3.5 g
200 (200 signals/220 grounds)	IT5HD-200S-BGA(**)	6.7 g
300 (300 signals/330 grounds)	IT5HD-300S-BGA(**)	12.8 g

** Receptacle will accept any available stacking height*

IT5 Mounting Receptacle

Contact Positions	Part Number	Weight
100 (100 signals/110 grounds)	IT5 M-100S-BGA(**)	2.7 g
200 (200 signals/220 grounds)	IT5 M-200S-BGA(**)	5.2 g
300 (300 signals/330 grounds)	IT5 M-300S-BGA(**)	9.9 g

** Receptacle will accept any available stacking height*

IT5 +1mm Mounting Receptacle

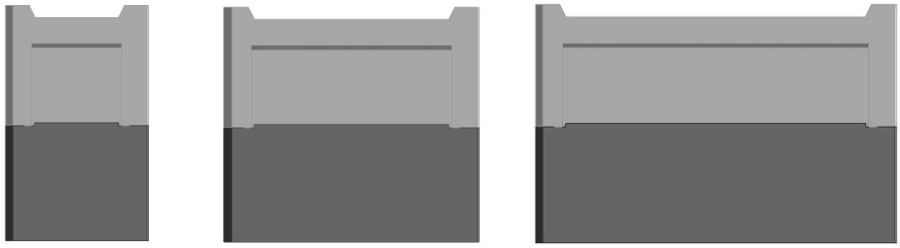
Contact Positions	Part Number	Weight
100 (100 signals/110 grounds)	IT5HM-100S-BGA(**)	3.5 g
200 (200 signals/220 grounds)	IT5HM-200S-BGA(**)	6.7 g
300 (300 signals/330 grounds)	IT5HM-300S-BGA(**)	12.8 g

** Receptacle will accept any available stacking height*

IT3/IT5 Interposer

IT3/IT5-100P-18H ***IT3/IT5-200P-18H*** ***IT3/IT5-300P-18H***

IT3/IT5-100P-38H ***IT3/IT5-200P-38H*** ***IT3/IT5-300P-38H***



IT3 Interposer Height Variation

Stacking Height	Contact Positions					
	100 pos. (100 signals)		200 pos. (200 signals)		300 pos. (300 signals)	
	Part Number	Weight	Part Number	Weight	Part Number	Weight
17 mm	IT3-100P-17H	6.5 g	IT3-200P-17H	10.8 g	IT3-300P-17H	15.1 g
18 mm	IT3-100P-18H	7.2g	IT3-200P-18H	12.1 g	IT3-300P-18H	17.0 g
20 mm	IT3-100P-20H	8.5 g	IT3-200P-20H	14.7 g	IT3-300P-20H	20.8 g
22 mm	IT3-100P-22H	9.9 g	IT3-200P-22H	17.3 g	IT3-300P-22H	24.6 g
25 mm	IT3-100P-25H	12.0 g	IT3-200P-25H	21.1 g	IT3-300P-25H	30.3 g
26 mm	IT3-100P-26H	12.7 g	IT3-200P-26H	22.4 g	IT3-300P-26H	32.2 g
28 mm	IT3-100P-28H	14.0 g	IT3-200P-28H	25.0 g	IT3-300P-28H	36.0 g
30 mm	IT3-100P-30H	15.4 g	IT3-200P-30H	27.6 g	IT3-300P-30H	39.8 g
32 mm	IT3-100P-32H	16.8 g	IT3-200P-32H	30.2 g	IT3-300P-32H	43.6 g
35 mm	IT3-100P-35H	18.7 g	IT3-200P-35H	34.6 g	IT3-300P-35H	49.7 g
38 mm	IT3-100P-38H	20.5 g	IT3-200P-38H	38.9 g	IT3-300P-38H	55.8 g
40 mm	IT3-100P-40H	22.3 g	IT3-200P-40H	40.6 g	IT3-300P-40H	58.8 g

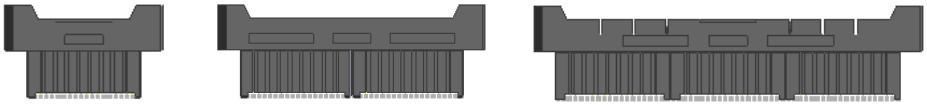
** Components listed in gray letters are under development*

IT5 Interposer Height Variation

Stacking Height	Contact Positions					
	100 pos. (100 signals)		200 pos. (200 signals)		300 pos. (300 signals)	
	Part Number	Weight	Part Number	Weight	Part Number	Weight
18 mm	IT5-100P-18H	6.9 g	IT5-200P-18H	11.7 g	IT5-300P-18H	17.2 g
22 mm	IT5-100P-22H	9.6 g	IT5-200P-22H	16.4 g	IT5-300P-22H	24.6 g
25 mm	IT5-100P-25H	12.6 g	IT5-200P-25H	22.3 g	IT5-300P-25H	33.2 g
28 mm	IT5-100P-28H	13.2 g	IT5-200P-28H	24.7 g	IT5-300P-28H	35.6 g
32 mm	IT5-100P-32H	17.5 g	IT5-200P-32H	33.8 g	IT5-300P-32H	49.1 g
35 mm	IT5-100P-35H	18.4 g	IT5-200P-35H	34.3 g	IT5-300P-35H	49.6 g
38 mm	IT5-100P-38H	22.2 g	IT5-200P-38H	41.7 g	IT5-300P-38H	60.0 g

IT3 Plug

IT3M-100P-15BGA ***IT3M-200P-15BGA*** ***IT3M-300P-15BGA***

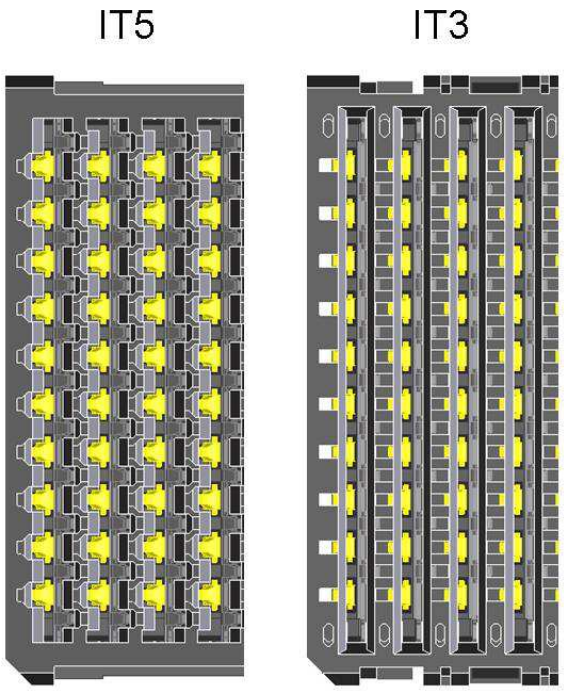


Stacking Height	Contact Positions	Part Number	Weight
14 mm	100 (100 signals/90 grounds)	IT3M-100P-14BGA(**)	6.5 g
	200 (200 signals/180 grounds)	IT3M-200P-14BGA(**)	12.2 g
	300 (300 signals/270 grounds)	IT3M-300P-14BGA(**)	17.7 g
15 mm	100 (100 signals/90 grounds)	IT3M-100P-15BGA(**)	7.0 g
	200 (200 signals/180 grounds)	IT3M-200P-15BGA(**)	12.7 g
	300 (300 signals/270 grounds)	IT3M-300P-15BGA(**)	18.2 g

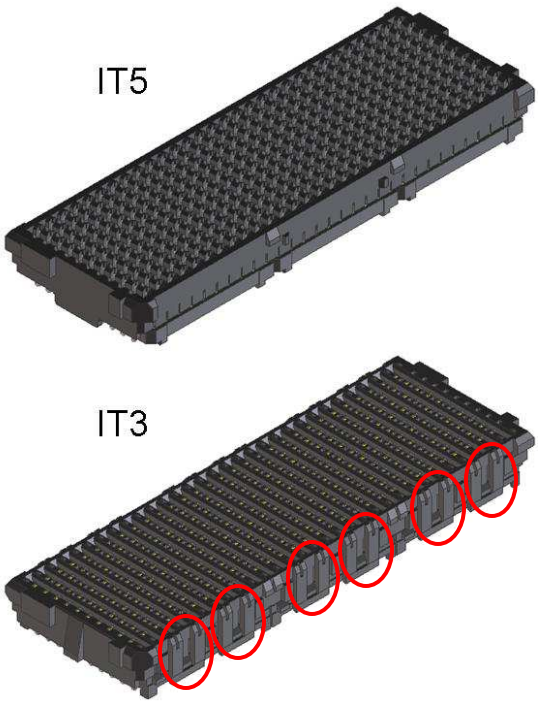
** Components listed in gray letters are under development*

2.2 Differences between IT3 and IT5 Connector Parts

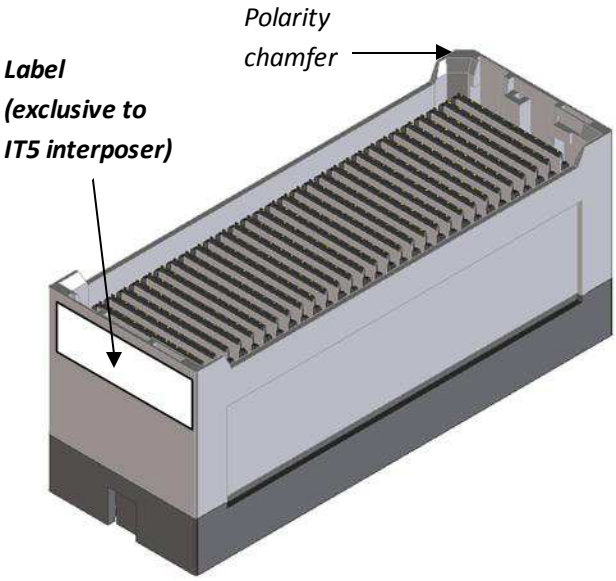
The differences between IT3 and IT5 receptacles are the following.



Housing design is different (its outlines defined in white for emphasis).



IT3 receptacles have latches on sides while IT5's do not.



The difference between IT3 and IT5 interposers is that, ***on IT5 interposer, a label is adhered on a shorter side away from the polarity chamfer*** *to indicate Hirose part number and other information.*

2.3 Part Number Designation

Receptacle

ITxxx - xxx S – BGA xx (xx)
(1)(2) (3) (4) (5) (6) (7)

Plug

ITxMx - xxx P – xxBGA xx (xx)
(1)(2) (3) (4) (9) (5) (6) (7)

Interposer

ITxxx - xxx P – xxH xx (xx)
(1)(8) (3) (4) (9) (6) (10)

(1) Series name

IT3: IT3; **IT5**: IT5

(2) Receptacle/Plug Type

ITxD-xxxS-BGA: Detachable (Mating) Receptacle
ITxDx-xxxS-BGA: Detachable (Mating) Receptacle (customized)
IT5HD-xxxS-BGA: +1mm Detachable (Mating) Receptacle
IT5HDx-xxxS-BGA: +1mm Detachable (Mating) Receptacle (customized)
ITxM-xxxS-BGA : Mounting Receptacle
ITxMx-xxxS-BGA: Mounting Receptacle (customized)
IT5HM-xxxS-BGA: +1mm Mounting Receptacle
IT5HMx-xxxS-BGA: +1mm Mounting Receptacle (customized)
IT3M-xxxP-xxBGA : Plug
IT3Mx-xxxP-xxBGA: Plug (customized)

(3) Contact Positions

100, 200, 300

(4) Connector

S : Socket; **P** : Plug

(5) BGA: Ball Grid Array

No further designation

(6) Package Specification

Blank: Standard; **xx**: Customized

(7) Material and Plating Specification of Receptacle and Plug

(37): Pb-free Solder: Sn (96.5) Ag (3.0) Cu (0.5)
Contact area: Gold(0.76 µm)+Ni(1.5 µm)
(Receptacle only) housing color: black
Note: IT5 receptacles are only available in Pb-free solder

(39): Pb-free Solder: Sn (96.5) Ag (3.0) Cu (0.5)
Contact area: Gold(0.76 µm)+Ni(1.5 µm)
(Receptacle only) housing color: gray
Note: IT5 receptacles are only available in Pb-free solder

(57): Eutectic Solder: Sn (63) Pb (37)
Contact Area: Gold(0.76 µm)+Ni(1.5 µm)
(Receptacle only) housing color: black
(59): Eutectic Solder: Sn (63) Pb (37)
Contact Area: Gold(0.76 µm)+Ni(1.5 µm)
(Receptacle only) housing color: gray

(8) Interposer type

Blank: Standard; **xx**: Customized

(9) Stacking Height (mm)

14, 15, 17, 18, 20, 22, 25, 26, 28, 30, 32, 35, 38, 40

(10) Plating Specification of Interposer

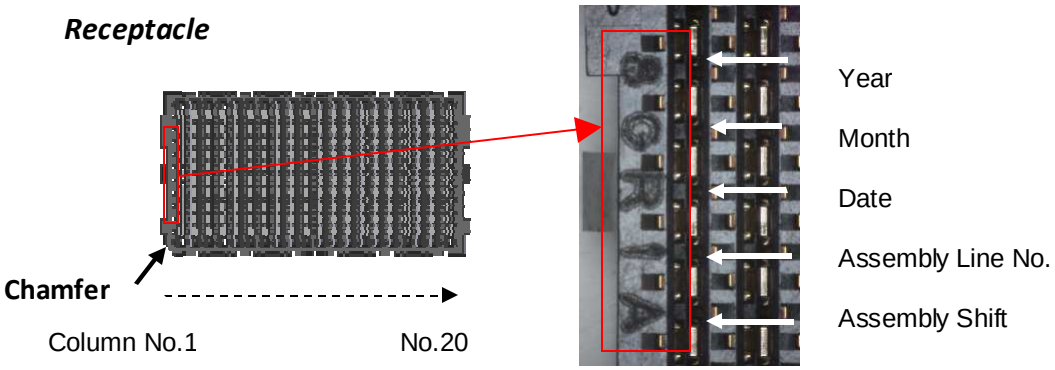
(03): Contact Area: Gold(0.76 µm)+Ni(1.5 µm)

***Customized IT3 interposers cannot mate with standard IT5 receptacles**

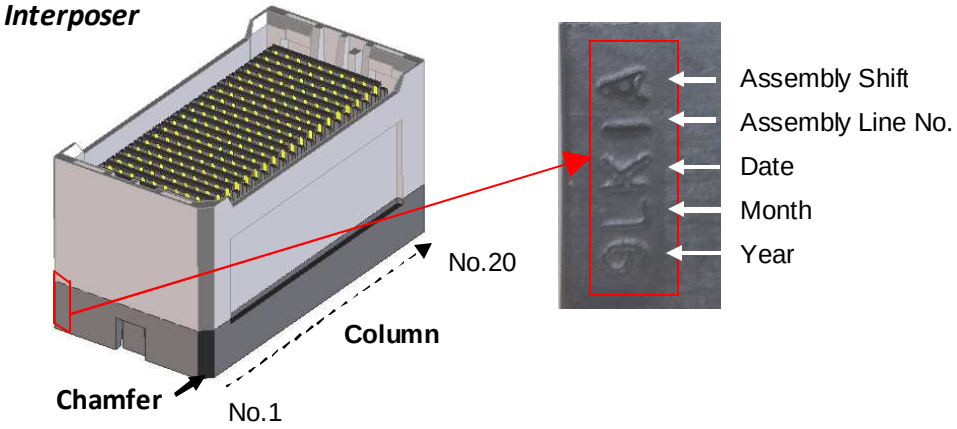
2.4 Manufacturing Lot Number

These configurations are the same with IT3 and IT5 connector parts.

Receptacle



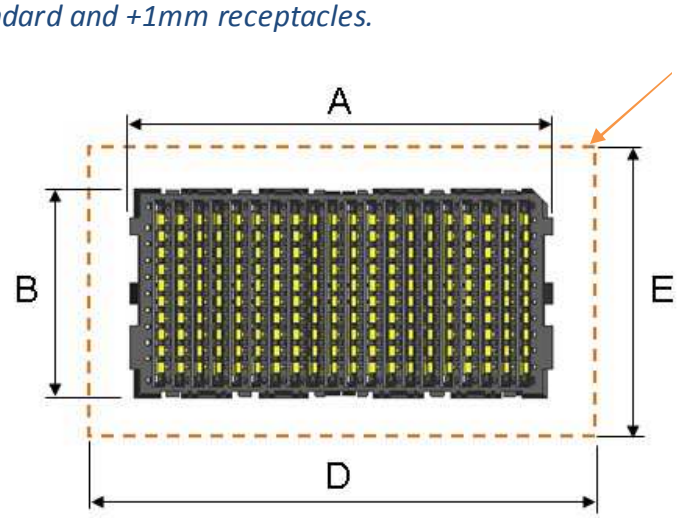
Interposer



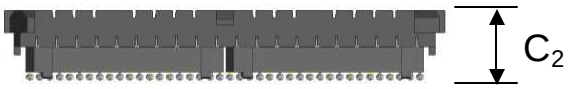
* Lot number indication may be subject to change.

2.5 Receptacle General Dimensions

These dimensions are the same with IT3 and IT5 standard receptacles. Dimensions A, B, D, and E are the same for standard and +1mm receptacles.



Shown: 200 position mounting receptade, IT3M-200S-BGA



Shown: 200 position mounting receptade, IT5HM-200S-BGA

		Contact Position		
		100	200	300
No. of Signal Contacts		100	200	300
No. of Ground Contacts (IT3 / IT5)		90 / 110	180 / 220	270 / 330
A	Receptacle Length	21	38.5	56
B	Receptacle Width	19.2	19.2	19.2
C_1	Regular Receptacle Height	6	6	6
C_2	+1mm Receptacle Height	7	7	7
D	Interposer/Plug Outline Length	24	41.5	59
E	Interposer/Plug Outline Width	21	21	21

**All dimensions shown are in mm*

Section 3 PWB Design Information

*The Hirose **IT3/IT5** connector's footprint is a staggered area array that allows space for easy via placement and signal routing between pads. Each row of I/O's alternates signal and ground interconnections. It is mounted to the board as a lightweight receptacle, and an interposer is used to connect to parallel PWBs at many different height options. Standoffs must be used in conjunction with the interposers to help reinforce the structure of the final multi-PWB assembly.*

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

Mating Receptacle

Interposer

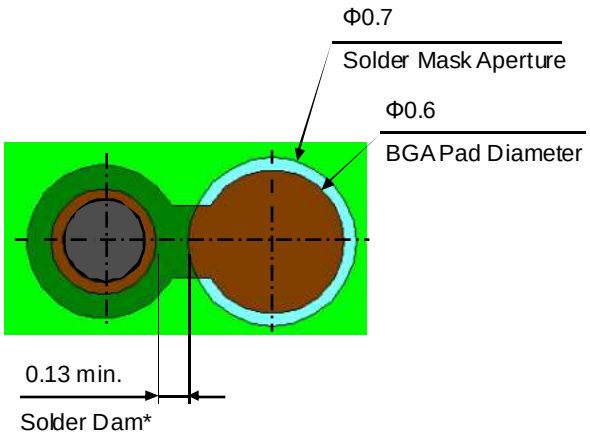
Mounting Receptacle

Shown: IT3-200P-26H

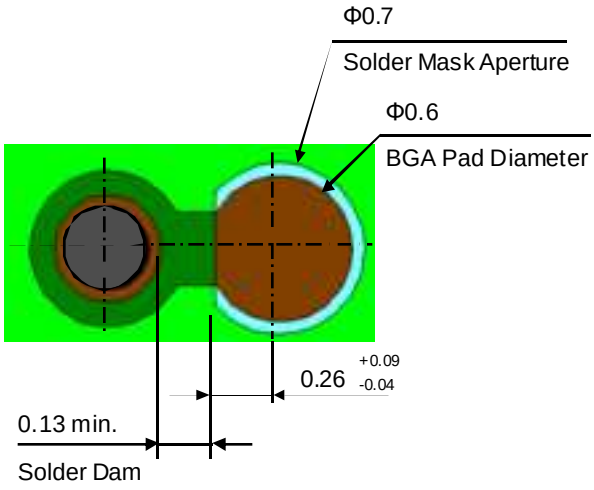
3.1 Footprint

Pad specification: 0.6mm diameter Non-Solder Mask Defined (NSMD), also known as *copper defined* or *metal defined,* pads are recommended. Recommended sizes and clearances are shown below.

HIROSE Recommended Pad Design



Acceptable Pad Design



* All dimensions shown are in mm

Through-via sizes will **depend on PWB thickness** and fabricator's capabilities. Vias should be placed far enough from the pad to ensure a **minimum** solder **dam width of 0.13mm**. **Circular openings** in the solder mask are **preferred,** but **D-shape openings** are **acceptable** if the minimum spacing requirement is met.

PWB pad finish is typically **Organic Solderability Preservative (OSP)** or **Hot Air Solder Level (HASL)**, but the component can also be used with Electroless Nickel-Immersion Gold (ENIG), Immersion Silver and Immersion Tin.

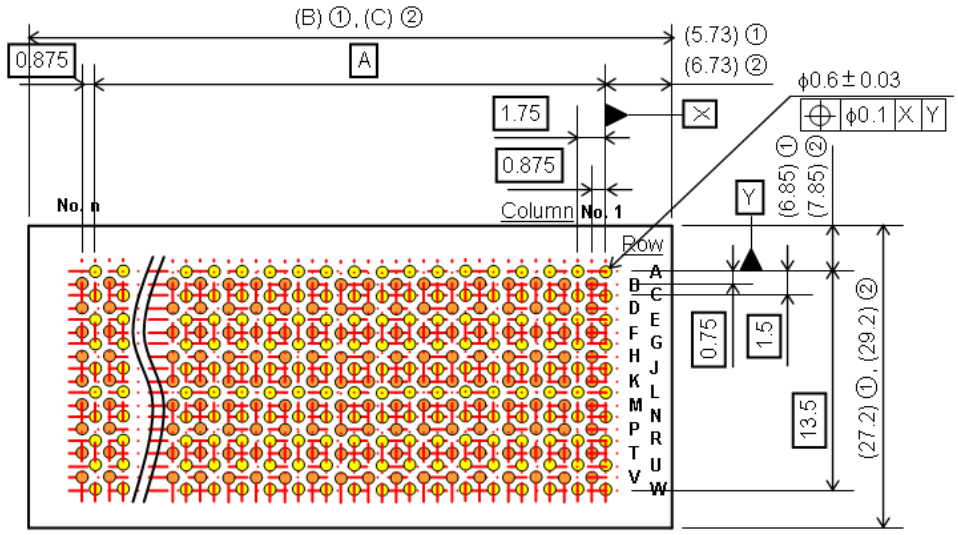
The **stencil apertures** should be **0.54mm circles**, concentric with the copper pads. This represents a 10% reduction from the diameter of the pad to compensate for typical variations in the assembly process.

The specified clearance, or **solder mask relief**, from the copper feature **is 0.05mm**.

***Precaution*: Verify fabricator capability. Solder mask registration** must be accurate to at least **0.05mm**. PWB fabricator's registration capability should be verified. Depending on thickness of PWB, fabricator's **aspect ratio** capabilities **for through vias** should also be verified.

Component Footprint and Contact Assignment

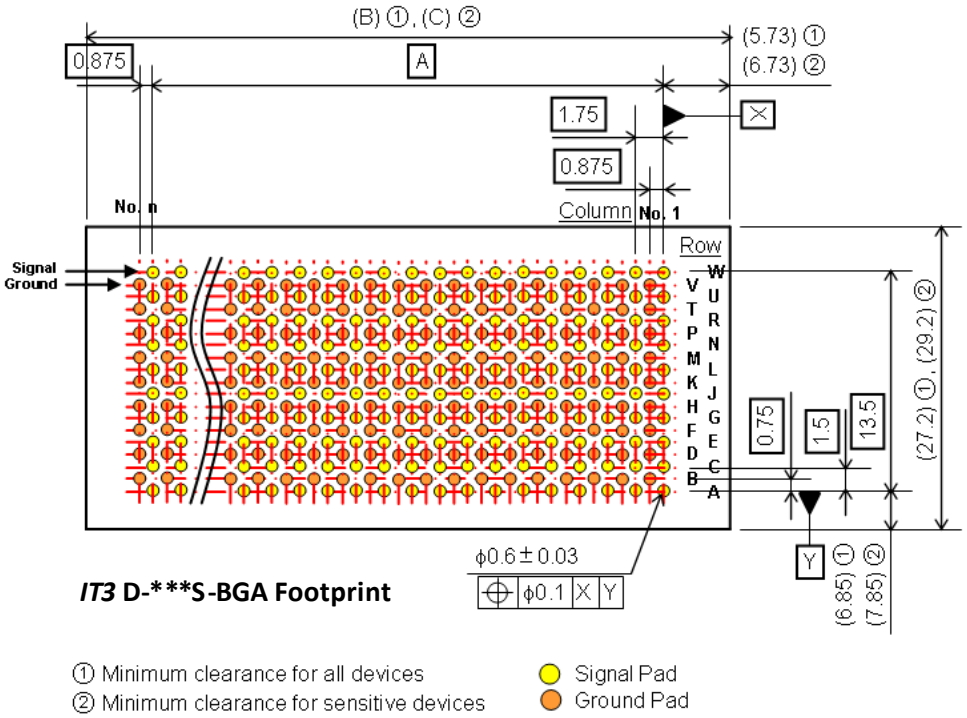
Mounting Receptacle – *IT3 M*



IT3* M-S-BGA Footprint**

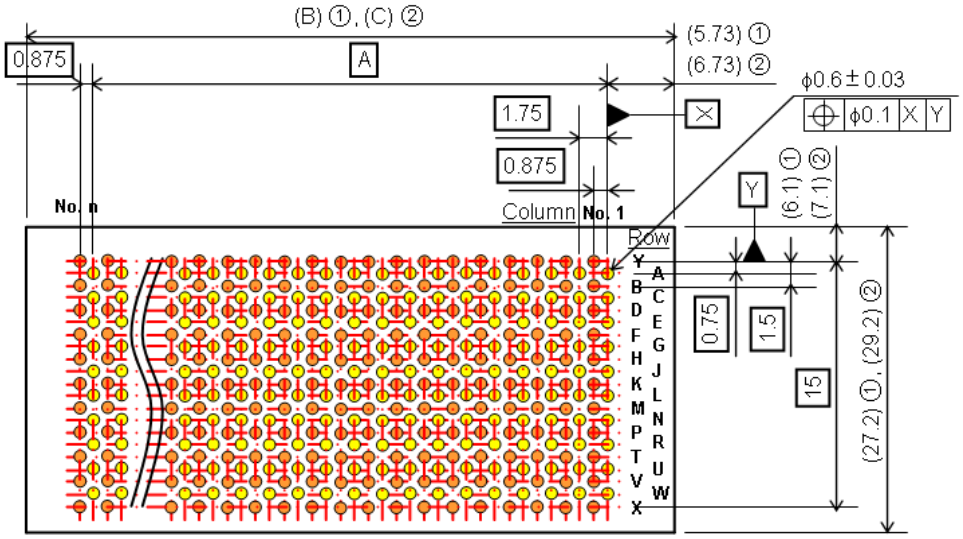
*** All dimensions shown are in mm.**

Mating Receptacle – *IT3 D*



IT3* D-S-BGA Footprint**

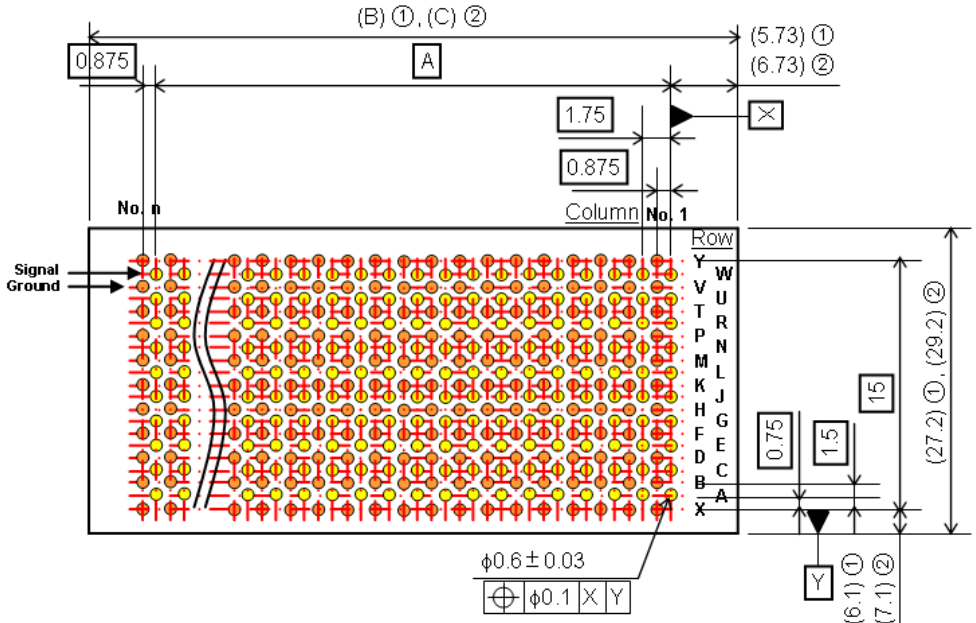
Mounting Receptacle – *IT5(H) M*



IT5* M-S-BGA Footprint**

*** All dimensions shown are in mm.**

Mating Receptacle – *IT5(H) D*



IT5* D-S-BGA Footprint**

① Minimum clearance for all devices
② Minimum clearance for sensitive devices

Signal Pad
Ground Pad

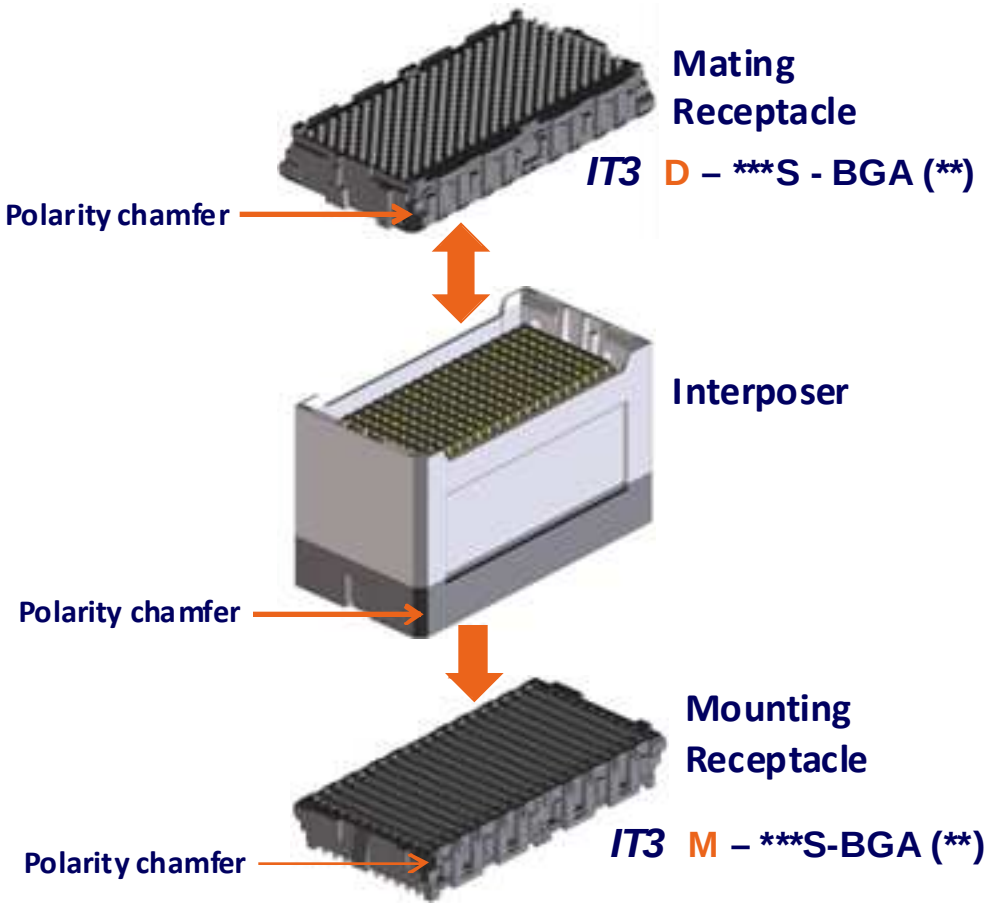
*** All dimensions shown are in mm.**

Dimension (mm)	100	200	300
A	15.75	33.25	50.75
B	28.10	45.60	63.10
C	30.10	47.60	65.10

The differences between IT3 and IT5 footprints are:

1. IT5 footprint has additional ground rows X and Y
2. IT5 footprint has narrower vertical minimum clearance for all devices and for sensitive devices.

Polarity: Each receptacle and interposer has **one corner chamfered** to insure proper orientation during assembly and installation. The corner with the **chamfer is nearest to pin A1**.

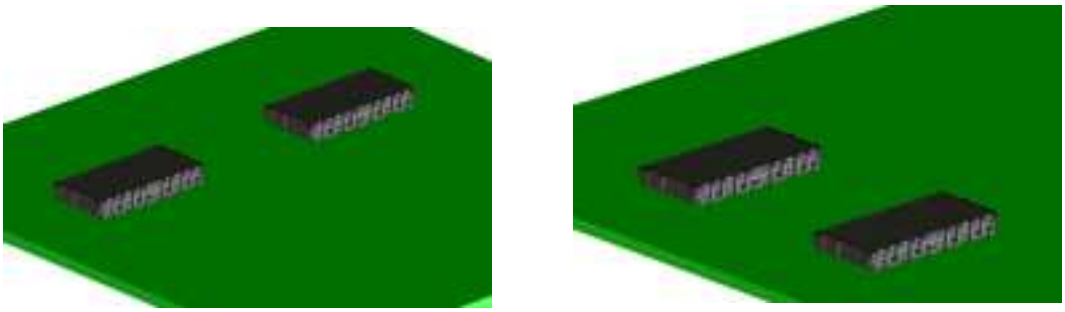


Annotation: For **visual inspection** purposes, **"Pin 1"** should be denoted on the silkscreen of the PWB by a **specific marking** (e.g. asterisk or other accepted symbol) near the A1 contact location and chamfer.

3.2 Multi-Connector Systems

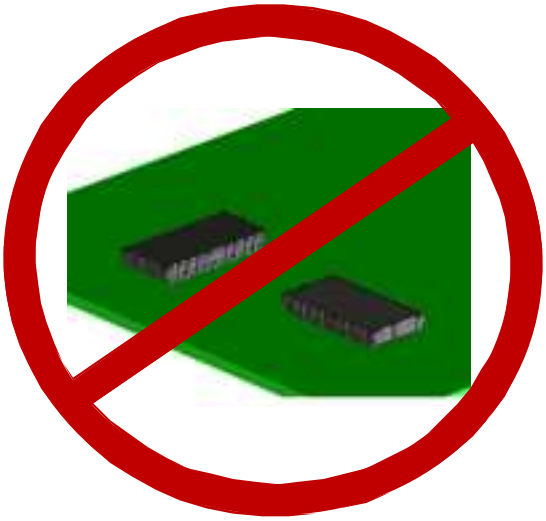
*The **IT3/IT5** connectors can be used singularly or in combination with other **IT3/IT5** connectors.*

If multiple connectors are used on the same PWB, they must be oriented in the same direction, as shown below:



Correct Orientations

It is not recommended to mix orientations:

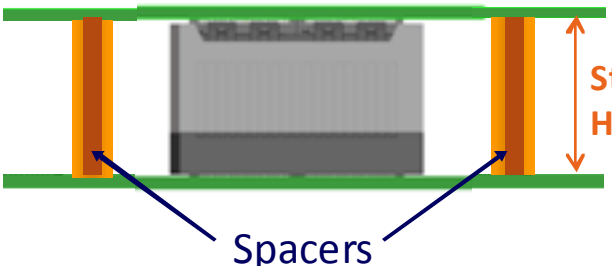


Do not mix orientations

It is not recommended to multiple mate with IT3 plugs for 2 piece solution.

3.3 Spacers

Spacers are required to support the PWB's and protect the BGA solder joints. Suggested spacer style is shown at lower right.

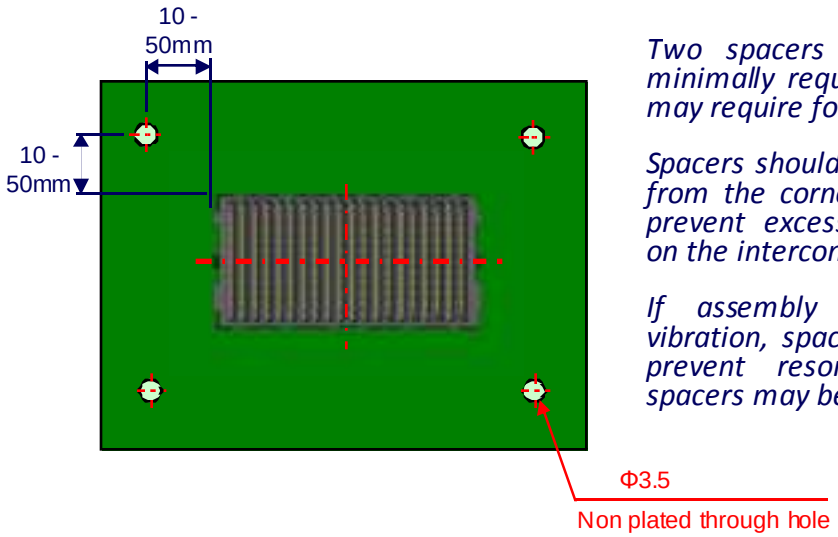


Spacer, male-male, M3 thread

The recommended spacer height corresponds to the interposer stacking height as shown in the chart below:

Stacking Height	Recommendecd Spacer Height	Stacking Height	Recommendecd Spacer Height
14 mm	14 +/-0.127 mm	29 mm	29 +/-0.127 mm
15 mm	15 +/-0.127 mm	30 mm	30 +/-0.127 mm
16 mm	16 +/-0.127 mm	32 mm	32 +/-0.127 mm
17 mm	17 +/-0.127 mm	33 mm	33 +/-0.127 mm
18 mm	18 +/-0.127 mm	34 mm	34 +/-0.127 mm
19 mm	19 +/-0.127 mm	35 mm	35 +/-0.127 mm
20 mm	20 +/-0.127 mm	36 mm	36 +/-0.127 mm
22 mm	22 +/-0.127 mm	37 mm	37 +/-0.127 mm
23 mm	23 +/-0.127 mm	38 mm	38 +/-0.127 mm
24 mm	24 +/-0.127 mm	39 mm	39 +/-0.127 mm
25 mm	25 +/-0.127 mm	40 mm	40 +/-0.127 mm
26 mm	26 +/-0.127 mm	41 mm	41 +/-0.127 mm
27 mm	27 +/-0.127 mm	42 mm	42 +/-0.127 mm
28 mm	28 +/-0.127 mm		

Recommended Spacer Location



Two spacers located diagonally are minimally required. Some applications may require four spacers.

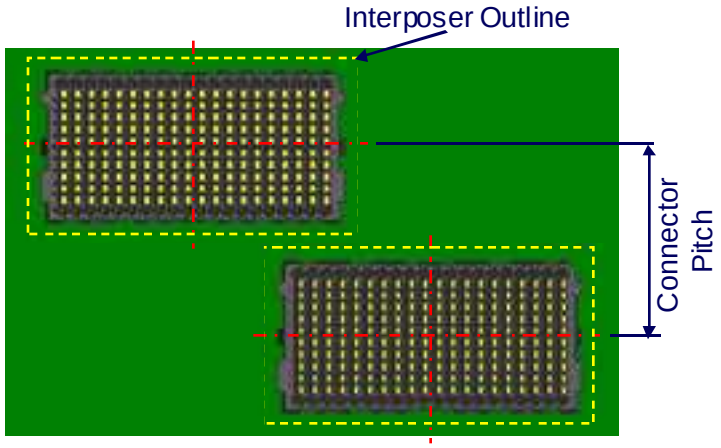
Spacers should be located 10 – 50 mm from the corners of the receptacles to prevent excessive mechanical loading on the interconnections.

If assembly will be subjected to vibration, spacers should be located to prevent resonance, and additional spacers may be required.

3.4 Clearance between Connectors

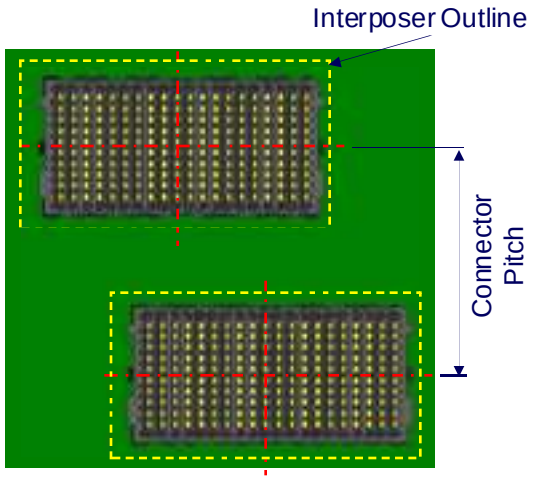
Parallel Mounting

If **overlap distance is less than half** the length of the connector:



Socket Combinations	Connector Minimum Pitch (mm)	Connector Maximum Pitch (mm)
All combinations	24.10	209.20

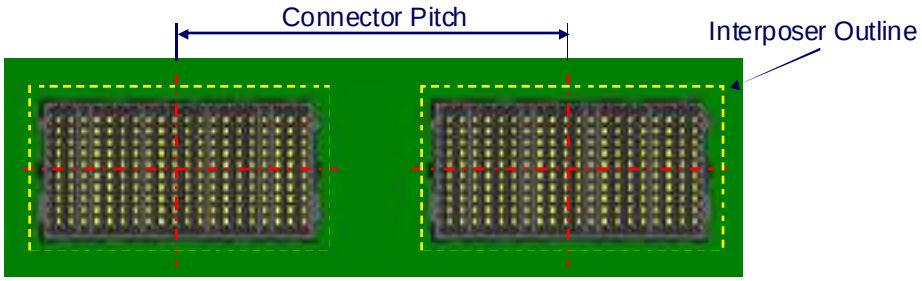
If **overlap distance is more than half** the length of the connector:



Socket Combinations	Connector Minimum Pitch (mm)	Connector Maximum Pitch (mm)
All combinations	31.00	209.20

Suggested clearances are based on accessibility to grip interposer for purposes of disassembly and field replacement.

Tandem Mounting

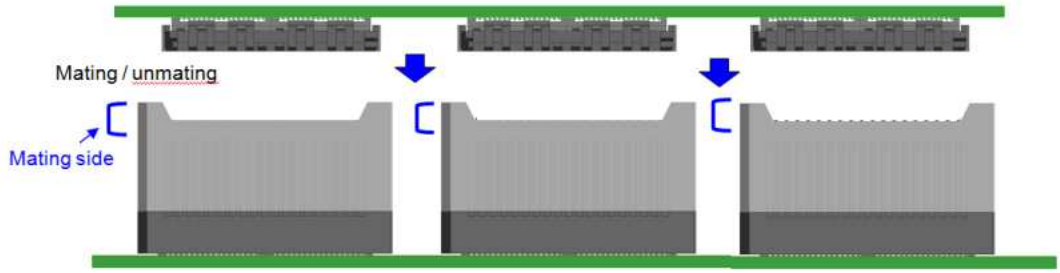


Socket Combinations	Connector Minimum Pitch (mm)	Connector Maximum Pitch (mm)
IT3/IT5-100pos + IT3/IT5-100pos	26.05	211.00
IT3/IT5-200pos + IT3/IT5-200 pos	43.55	228.50
IT3/IT5-300pos + IT3/IT5-300 pos	61.05	246.00
IT3/IT5-100pos + IT3/IT5-200 pos	34.80	219.75
IT3/IT5-100pos + IT3/IT5-300 pos	43.55	228.50
IT3/IT5-200pos + IT3/IT5-300 pos	52.30	237.25

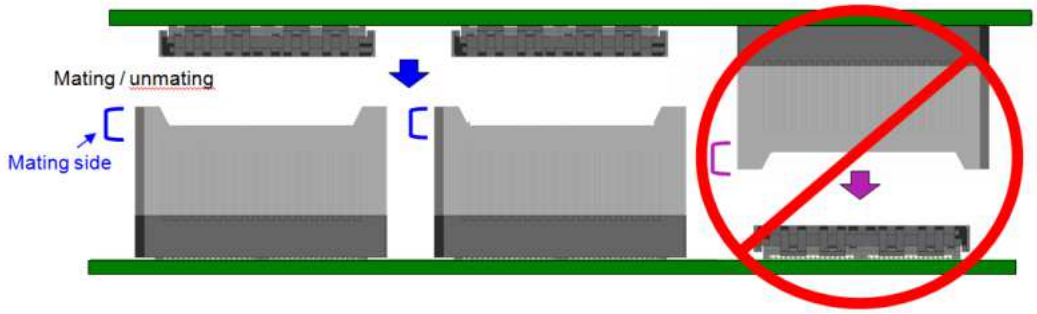
3.5 Interposer Direction

Do not mix mating and mounting receptacles on the same PWB.

All interposers must engage in the same direction, as shown below:



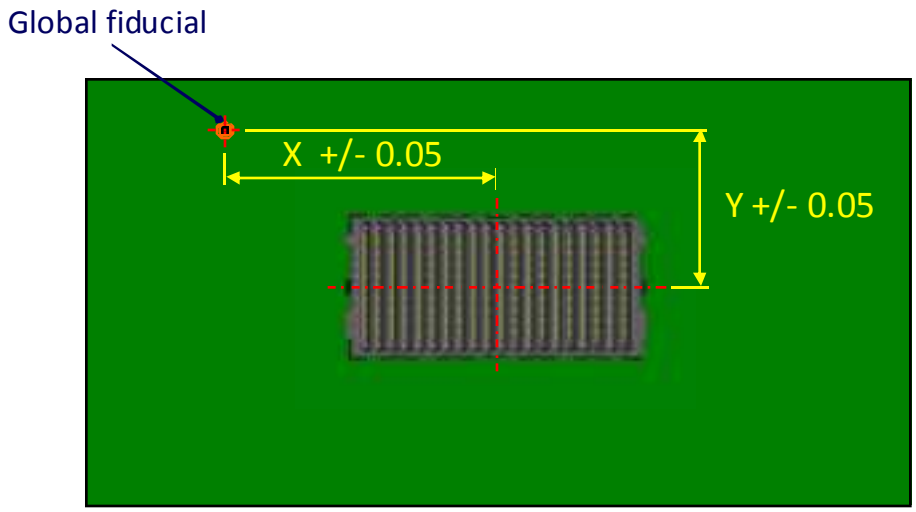
Correct Method – all connectors mate in same direction



Incorrect Method – connectors mate in different directions

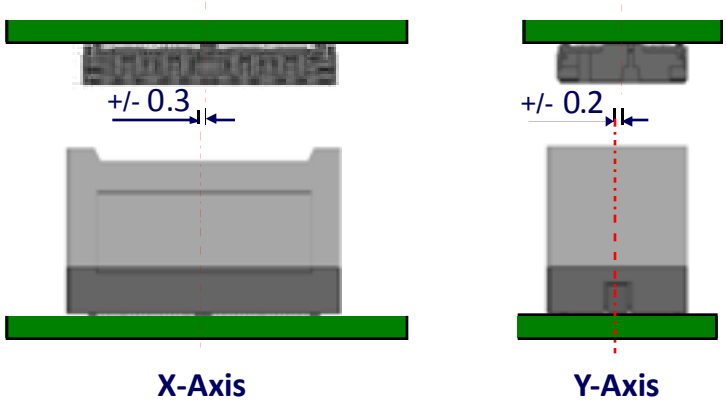
3.6 Alignment Tolerances

Mounting Tolerances of ± 0.05mm are required for robust SMT assembly and to ensure proper mating fits in cases of multiple connectors:



*All dimensions shown are in mm.

Mating tolerances: Due to its 3-piece design, the IT3/IT5 connector system can accept mating tolerances of up to ±0.3mm tolerance in the X-axis and up to ±0.2mm in the Y-axis.



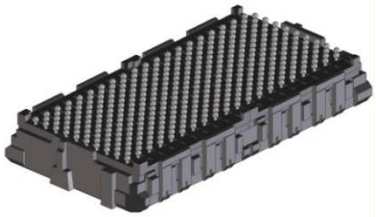
* All dimensions shown are in mm

Section 4 Introduction to Assembly

*The Hirose **IT3/IT5** BGA Connector System is fully compatible with a wide range of SMT equipment, and can be assembled on nearly any production line without requiring special tooling or machinery. The connector receptacle's printing and placement characteristics are similar to 1.0mm BGA devices, and its low profile, open body design opens the reflow window. It is available in tin-lead and lead-free configurations for IT3 and lead-free configurations for IT5, with I/O counts ranging from 190 to 630.*

The components of the system confirm to the packaging requirement of ISTA 3A (standard). They are categorized in the MSL2A of IPC/JEDEC J-STD-020B, and are recommended to be used within four weeks after being unpacked in the office environment defined by JEDEC.

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.



***IT3* Mating Receptacle**

Section 5 Stencil Printing

The Hirose IT3/IT5 BGA connector has print characteristics similar to 1.0mm BGA device. Its pads are 0.6mm diameter on a staggered 1.5 x 1.75mm grid. It requires no special aperture designs or stencil thicknesses, and can be successfully printed in any process that is capable of printing 1.0mm BGA's.

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

5.1 Solder Paste Selection

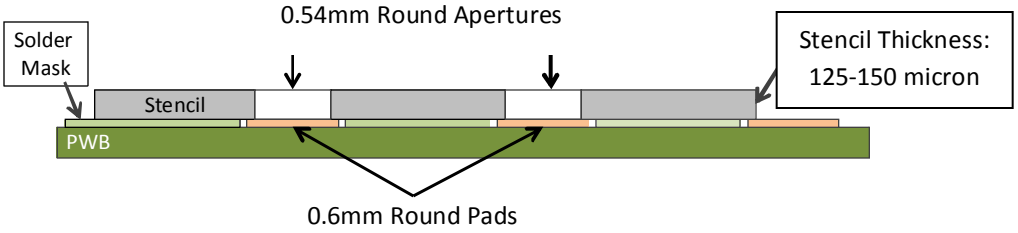
The ***IT3/IT5*** connector receptacle is compatible with a broad range of solder pastes. It can be used with **no-clean or water washable** products that include **Type 3** (particle size 25-45 microns) **or Type 4** (particle size 20-38 microns) powders. IT3 connectors are available in both **tin-lead and lead-free** configurations and IT5 connectors are available in **lead-free** configurations. The user should assure that the solder alloy used in the assembly process is compatible with the appropriate component configuration.

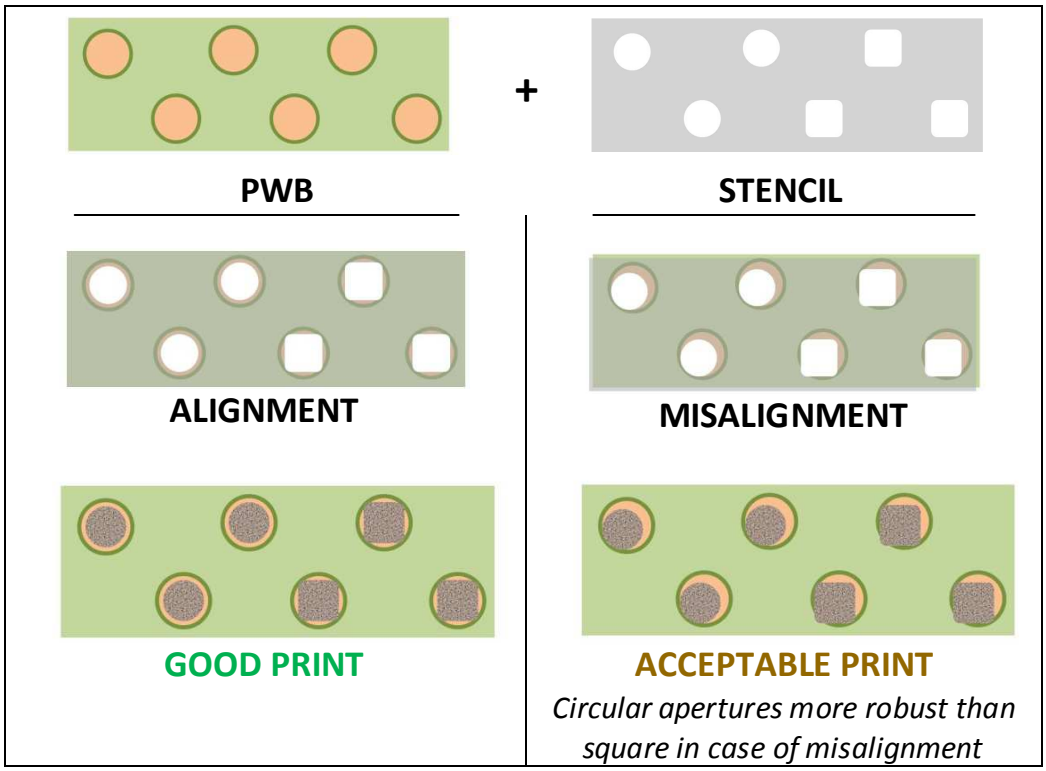
5.2 Stencil Design

The suggested PWB pad design is non-solder mask defined 0.6mm diameter circles, patterned as shown in section 3.1. A **0.54mm circular stencil aperture is recommended** to correspond to the 0.6mm pad. The diameter of the aperture is intentionally reduced to compensate for typical misalignment between the stencil's apertures and the PWB's pads, which can affect print quality. 0.54mm square apertures may also be used; if so, their corners should have radius of at least 0.1mm.

5.3 Effect of Alignment

Stencil **thicknesses of 125 to 150 microns are best**; thicknesses of 100-124 microns and 151-175 microns are acceptable. Thicknesses less than 100 microns or greater than 175 microns are not recommended.

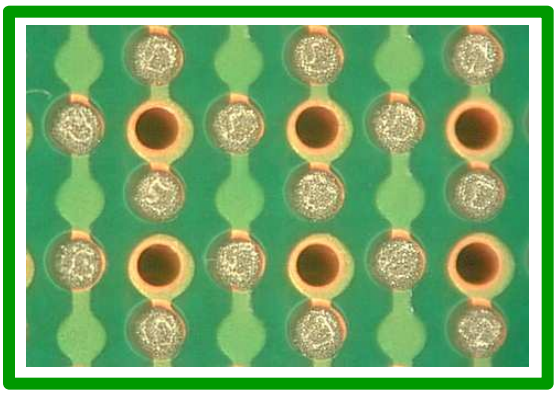




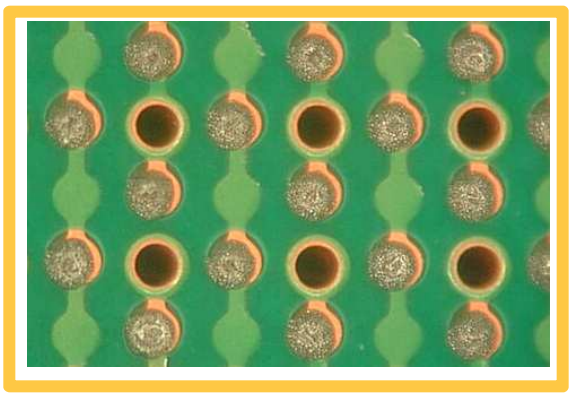
5.4 Paste Print Quality

Aperture diameters can be enlarged to accommodate other process considerations, particularly if the printing process is well controlled. Solder deposits should be of uniform size and shape, with good definition. Excess flux or paste between the pads is not acceptable

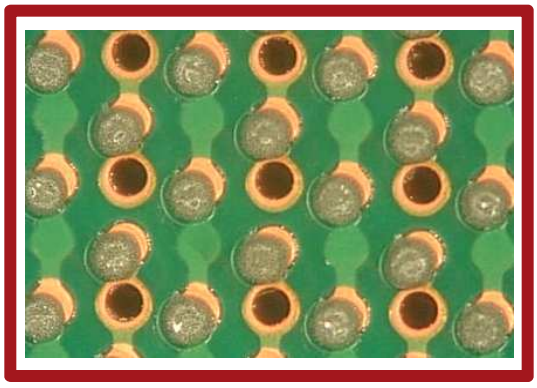
Solder Paste Print Comparison



Preferred



Acceptable



Unacceptable

Precautions: Solder paste print quality is a very important factor in achieving a high quality assembly process. If preferred or acceptable print quality is not obtained, refer to section 11, **Tips for SMT Assembly.**

Section 6 Pick and Place

*All Hirose **IT3/IT5** standard receptacles are 6mm tall and IT5H +1mm receptacles are 7mm tall. They require no special modifications to vision or placement routines to accommodate differrent component heights if using all standard or +1mm receptacles. They are shipped in JEDEC trays with pick and place tape attached on 100 and 200 position receptacles, and metal cap attached on 300 position receptacles. Orientation is indicated on the receptacles and trays by a chamfered corner.*

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

6.1 Packaging

Since receptacles have solder balls, receptacle trays will be sealed inside a bag with deoxidizing agent and desiccant. Please do not mount these receptacles after 4 weeks of opening the bags according to J-STD-0208, MSL2A in IPC/JEDEC standards. Receptacles comply with JEDEC standards (JEDEC PUBLICATION 95, DESIGN GUIDE 4.10).

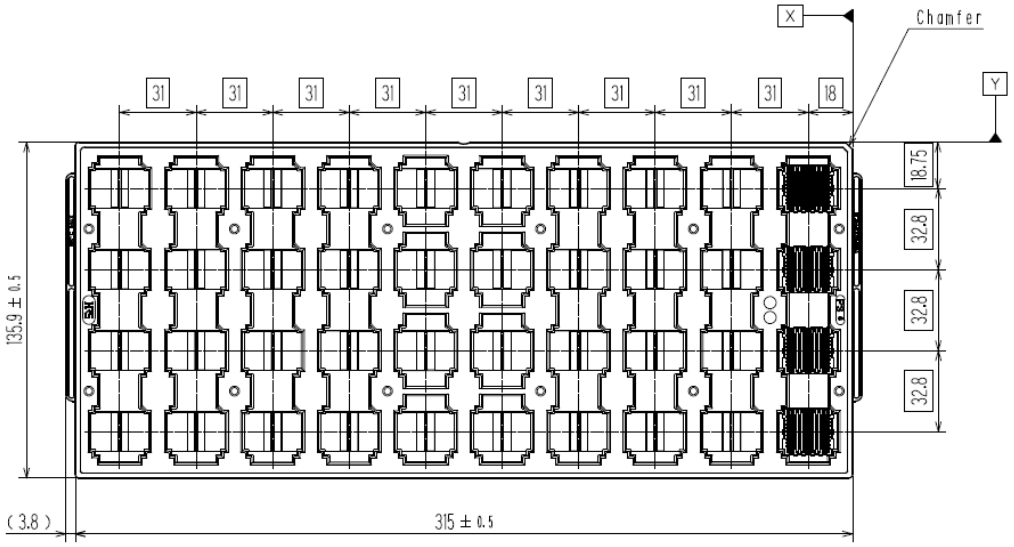
The expiration time for receptacles within close bags is 12 months (or one year) from the delivery date when the effect of deoxidizing agent and desiccant is gone. If an operator repackages and reseals receptacles within 12 hours, that will allow receptacles to get back into store-able condition and be stored for total of 12 months. The desiccant should have minimal amount of exposure to the air during repackaging. The bag provided is rated for 12 months.

Representative packaging for IT3 receptacles

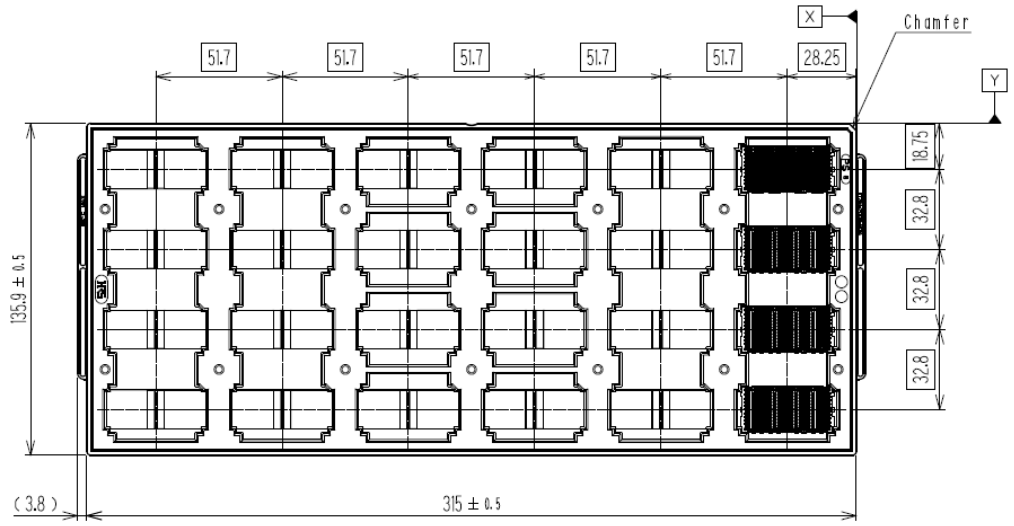
Representative packaging for IT5 receptacles

6.2 Media Trays

IT3/IT5 receptacle components are shipped **with pick and place tape attached**, in JEDEC hard trays as shown below.

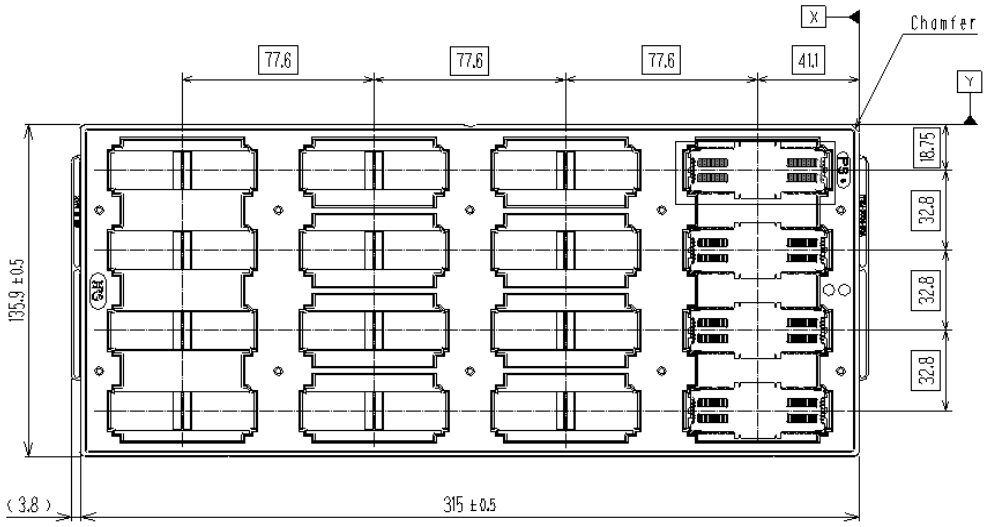


JEDEC Tray for *IT3/IT5 100 Position* Receptacles



JEDEC Tray for *IT3/IT5 200 Position* Receptacles

* All dimensions shown are in mm.



JEDEC Tray for *IT3/IT5 300 Position* Receptacles

* All dimensions shown are in mm.

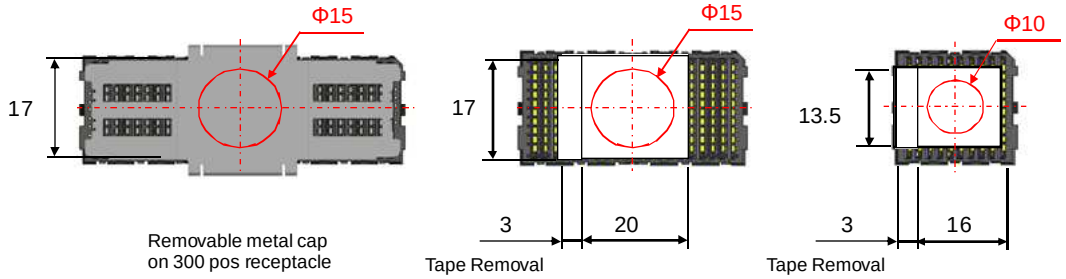
The difference between IT3 and IT5 receptacle trays is that IT5 trays are gray-colored.

6.3 Pre-bake

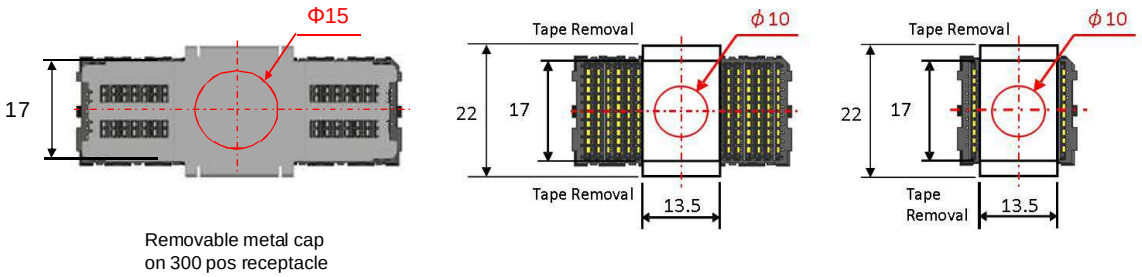
The connector body materials do not absorb water from the atmosphere; therefore **pre-bake is not required** for IT3 and IT5 receptacles.

6.4 Pick-Point

Pick locations and dimensions for IT3 and IT5 receptacles are shown below. Pick points are located at the geometric centers of receptacles.



For IT3 series, the 300 position receptacle has a metal cap that is easy to remove after reflow, as shown in Section 7.10. The 200 and 100 position receptacles have adhesive tape which has a 3mm margin without adhesive for easy removal.



Removable metal cap
on 300 pos receptacle

For IT5 series, the 300 position receptacle has the same metal cap as the IT3 one. The 200 and 100 position receptacles have adhesive tape which has margins on top and bottom for removal.

6.5 Component Weight

Contact Positions	Part Number	Solder Ball Material	Weight
100	IT3M/D-100S-BGA(37/39)	Pb-free (SAC 305) solder	2.5 g
	IT3M/D-100S-BGA(57)	Eutectic SnPb solder	
	IT5M/D-100S-BGA(37/39)	Pb-free (SAC 305) solder	2.7 g
	IT5HM/HD-100S-BGA(37/39)	Pb-free (SAC 305) solder	3.5 g
200	IT3M/D-200S-BGA(37/39)	Pb-free (SAC 305) solder	4.7 g
	IT3M/D-200S-BGA(57)	Eutectic SnPb solder	
	IT5M/D-200S-BGA(37/39)	Pb-free (SAC 305) solder	5.2 g
	IT5HM/HD-200S-BGA(37/39)	Pb-free (SAC 305) solder	6.7 g
300	IT3M/D-300S-BGA(37/39)	Pb-free (SAC 305) solder	9.1 g
	IT3M/D-300S-BGA(57)	Eutectic SnPb solder	
	IT5M/D-300S-BGA(37/39)	Pb-free (SAC 305) solder	9.9 g
	IT5HM/HD-300S-BGA(37/39)	Pb-free (SAC 305) solder	12.8 g

6.6 Vision and Alignment

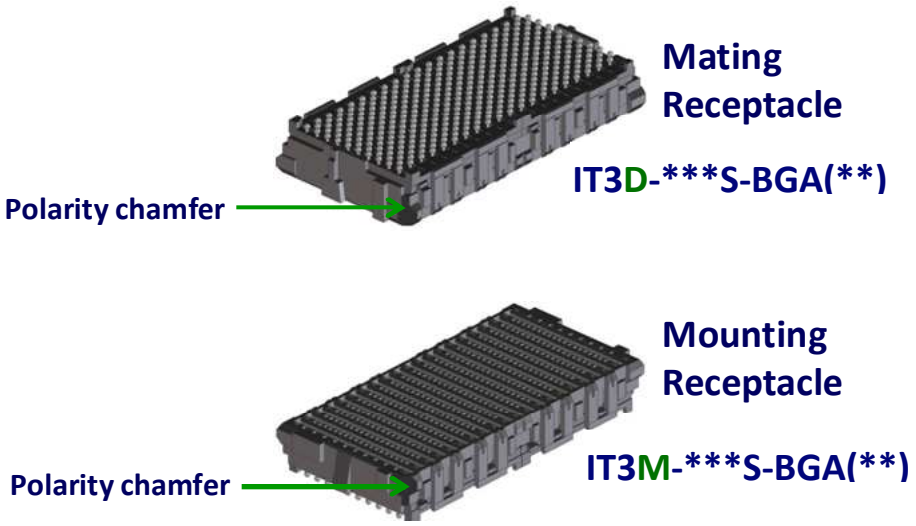
Pick and place vision systems should **inspect all balls** and use best practices available for image capture/analysis and centering.

6.7 Placement Force

Placement force should be adequate to firmly position component in solder paste without grossly deforming deposits. **For best results, balls should be approximately ⅓ to ½ the depth of the solder paste.** Consult placement equipment manufacturer for specific guidelines regarding BGA mounting.

6.8 Orientation Indicator

The receptacle's polarity is indicated by a chamfer on one corner, as shown below:



Section 7 Reflow Processing

*The low profile design of the Hirose **IT3/IT5** connector receptacles has relatively low thermal mass and provides open pathways to allow efficient heat transfer to the interconnections. The package provides low temperature differentials between interior and exterior solder joints, and between the interconnections and the package body. Because all standard receptacles are the same height and mass, no special profiling considerations are required to accommodate different stacking height requirements of the final assemblies. Lead-free BGA balls are SAC305 alloy.*

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

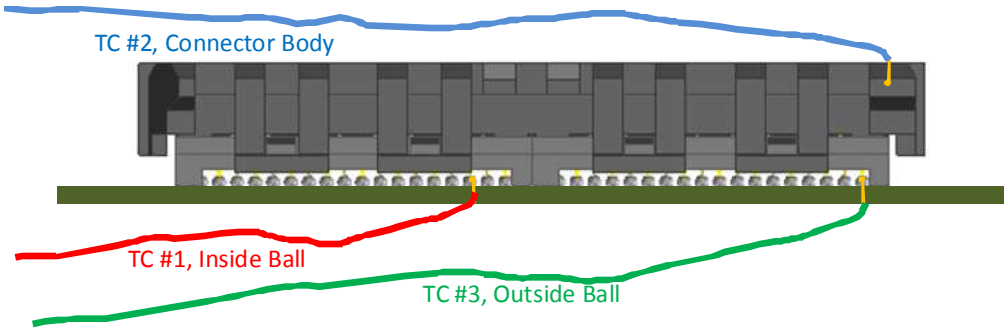
7.1 Instrumentation

The reflow process should be recorded using a digital data logger with thermocouples attached to key locations on the PWB and connector.

7.2 Thermocouple Attachment

A **minimum of two thermocouples** should be used on the IT3/IT5 connector receptacle. One should be located on an **inside ball**, preferably a ground connection (coolest solder joint), and one on the **connector body**. A **third thermocouple** may be placed on an **outside ball,** preferably a signal connection (hottest solder joint).

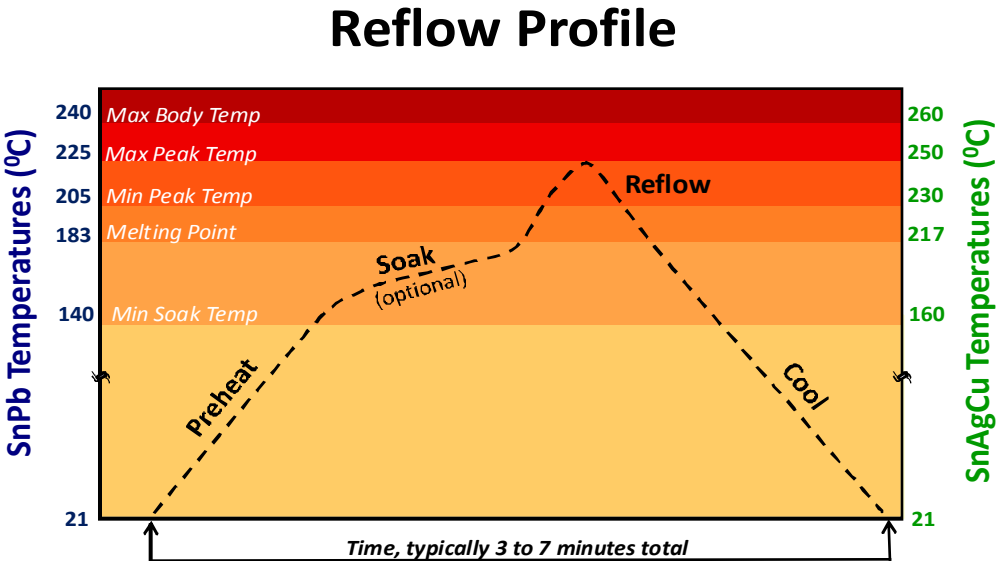
Note: It is **very important** to attach thermocouples to balls by drilling through PC board and mounting from bottom side as shown below:



7.3 Reflow Profile Considerations

Parameters	Tin-Lead	Lead-Free	Comment
Preheat Ramp Rate	2 - 3°C/sec	2 - 3°C/sec	Other components may limit ramp rate to 2°C/sec
Soak Time	0 - 120 sec	0 - 120 sec	Soak requirements determined by board design, oven capability, and paste activation requirements
Soak Temperature	140 - 180°C	160 - 215°C	Caution - "oversoaking" may exhaust flux and affect soldering
Peak Reflow Temperature	205 - 225°C	230 - 250°C	Cooler peak temperatures may require longer TAL's
Time Above Liquidus (TAL)	30 - 90 sec	45 - 120 sec	Shorter TAL's may require higher peak temperatures
Cooling Rate	>6°C/sec	>6°C/sec	Faster cooling rates produce finer grain structures and smoother joint appearances
Maximum Package Body Temperature (T)	240°C	260°C	Open body design allows for low delta T between package and solder joint
Maximum Delta T between Body and PWB at Liquidus	10°C	10°C	Standard practice is easy to achieve with open body design
Package Body Exposure Limit at Maximum Temperature	5 sec	5 sec	Adjust profile if maximum exposure limit is approached or exceeded

7.4 Suggested Thermal Profile Ranges



Different solder pastes have different thermal performance characteristics. Consult with paste manufacturer for optimum profile settings.

7.5 Other Important Factors

Check thermal exposure limits of PWB laminate if processing with lead-free solder.

7.6 Nitrogen Environment

The use of nitrogen to inert the reflow process *can*:

1. improve solder wetting by limiting oxidation on metal surfaces
2. allow lower peak temperatures and/or lower times above liquidus
3. improve the cleanability of flux residues

The benfits of using nitrogen increase as oxygen levels are decreased; optimum oxygen levels between 100 and 1500 ppm must be determined by the assembler.

Generally speaking, nitrogen inerting opens the reflow window for most elctronic components. The **use of nitrogen** in the reflow process will likely improve the soldering preformance of the ***IT3/IT5*** connector receptacle, but it **is not required**.

7.7 Double-sided and Inverted Reflow

The ***IT3/IT5*** **receptacles are compatible with the double-sided and inverted reflow** soldering. Each receptacle has a low profile and a relatively light weight. The weight-to-ball ratio for receptacles is less than 0.025 g/ball, and they have been successfully assembled in inverted positions in production environments. While the IT3 plugs are also compatible with the double-sided reflow soldering, please be cautious of the inverted reflow soldering with the plugs. Thermoset plastic can be used to bond the standoffs of a plug to a PCB, thus creating extra adhesion to prevent the plug from potentially falling out during the inverted reflow soldering.

7.8 Mixing Tin-Lead and Lead-Free

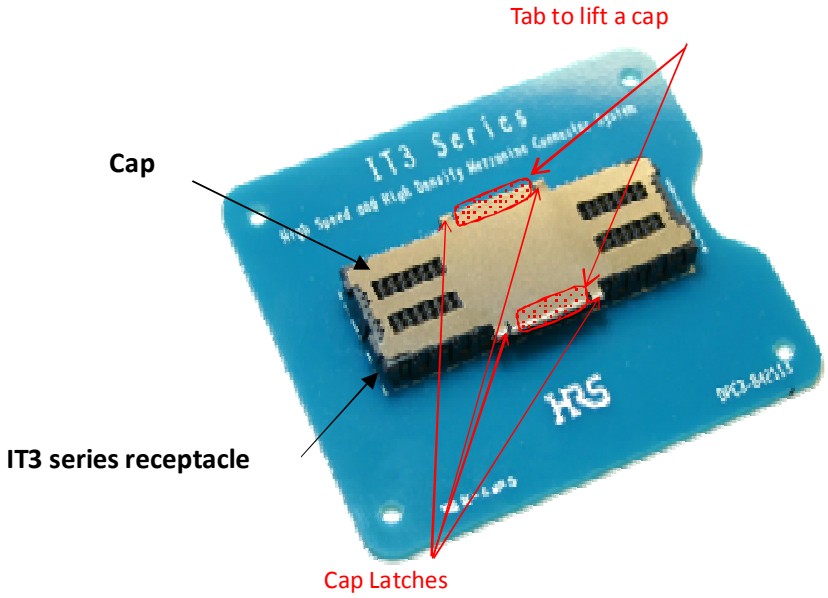
The ***IT3*** connector receptacle is available in both tin-lead and lead-free configurations and **IT5** connector receptacles is available in lead-free configuratios. Therefore, **mixed metals systems should not be considered**. However, if a situation requires mixing alloys on an assembly, it is **not advisable to use tin-lead components in lead-free processes**; large solder voids can result. **Lead-Free components may be used in tin-lead processes**, but must reach **peak temperatures above 221°C** to ensure full collapse and homogeneous joint microstructure. *If mixed metals systems are under consideration, please contact product designers for approval prior to assembing connector receptacles.*

7.9 Alternative Alloys

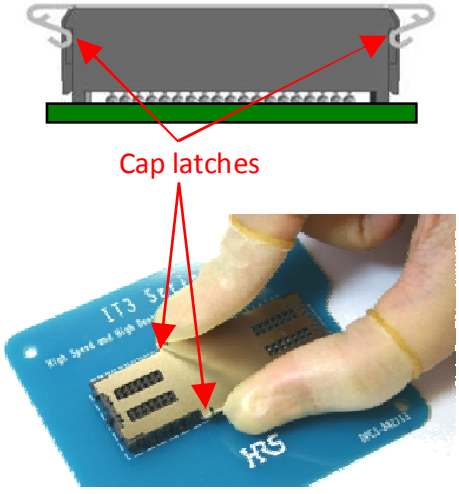
BGA ball alloys other than SnPb or SAC305 may be available. Contact Hirose for more information.

7.10 Cap Removal

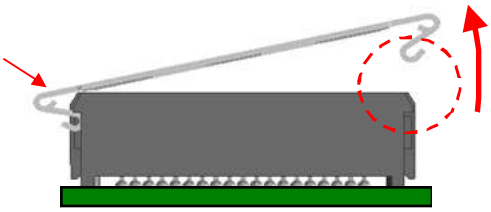
The metal cap is secured onto the 300 pos receptacle with four small latches that are located on either side of the lifting tabs.

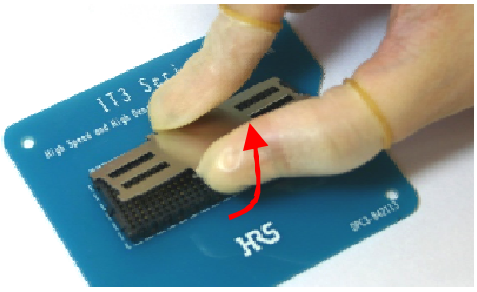


To remove the cap, hold it by its removal tabs:

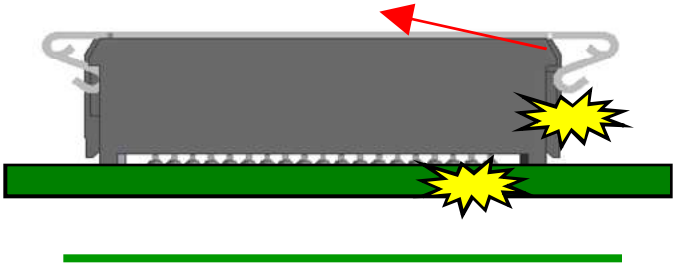


Lift upward on one side while holding the other side in place.





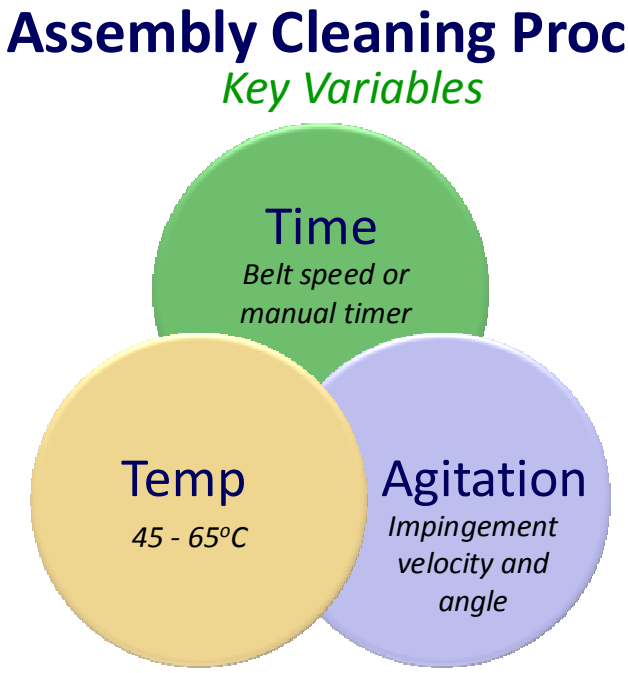
Precaution: It is important to **"roll" the cap off the connector** gently. It is designed to separate very easily with this method. Pushing the latch into the connector will not cause separation and may damage the connector body and/or solder joints.



Section 8 Cleaning

*The **IT3/IT5** connector system receptacle's post-reflow standoff height of 0.8 mm and open body design make it very easy to clean solder paste residues, even with modest cleaning processes.*

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.



8.1 Water-Washable Solder Process

For water washable solder pastes, washing action is a function of **Temperature, Time and Agitation**. **Consult with solder paste supplier for** optimum water **temperature**, as some residues clean better at lower temperatures (45 – 50°C) and others clean better at higher temperatures (60 – 65°C). Cleaning **time is a function of belt speed** for in-line cleaning machines, and can be set manually for most batch cleaners. **Agitation** is the mechanical scrubbing action of the water, and can be controlled by changing the flow rate, pressure, number of nozzles or angle of the water delivery system, depending on the type of cleaning machine used. For most BGA devices, a shallow impingement angle is required to help direct the cleaning fluid under the package body and aid the cleaning process. The ***IT3/IT5*** connector's open body design allows fluid flow from any direction and does not require shallow impingement angles to ensure fluid penetration.

8.2 Saponifiers

All solder pastes that are classified as water soluble or water washable may be cleaned effectively with hot water alone, but **their solubility greatly improves if saponifiers are added** to the cleaning bath. Consult with solder paste supplier for compatible cleaning agents and appropriate concentrations. Prior to using any type of cleaning chemistry or solvent, review the manufacturer's instructions for use and its effects on metals or plastics, and consult with the cleaning equipment manufacturer for system compatibility.

Precaution: When using water soluble solder pastes, it is extremely important to completely remove all flux residues from the assembly. Water soluble fluxes contain organic acids that are active at room temperature, and other ionic substances that can cause corrosion and/or electrochemical migration. **Incomplete removal of water soluble flux residue can result in premature failure of the assembly in its service environment.**

8.3 No-Clean Solder Process

No-Clean solder pastes are designed to be left on the circuit board. If reflowed properly, the residues will not interfere with circuit function. Many no-clean fluxes contain rosin, which is not water soluble**. If cleaning of no-clean flux residue** is desired, **cleaning agents must be used**. Contact solder paste supplier for compatible cleaning agents and appropriate concentrations. As with any cleaning chemistry, review the manufacturer's instructions for use and its effects on metals or plastics, and consult with the cleaning equipment manufacturer for system compatibility

The ***IT3/IT5*** receptacle has passed compatibility testing with IPA

Note: *In high frequency applications (>30 GHz), some no-clean flux residues can interfere with signal integrity and may require cleaning. Contact solder paste manufacturer for information on high frequency compatibility.*

Precaution: When using organic solvents or water type cleaners, any residual flux or cleaning agent that remains on the assembly can deteriorate the electrical performance of the connector and/or cause permanent damage. **PWB and components should be completely clean and dry before they are energized.**

8.4 Cleanliness Testing

Ionic cleanliness **cannot** be confirmed visually. The absence of visible residues **does not** indicate complete cleanliness. Ionograph or ion chromatography tests must be used to verify cleanliness and fitness for use.

The **ionograph** measures **total** ionic contamination over the **entire surface** of the PWB, and is typically used as a process indicator to monitor cleaning processes in production environments.

Ion chromatography measures **localized** ionic contamination in **specific areas** of the PWB, and is typically used during development efforts to verify the cleanability of a soldering product, the effectiveness of a cleaning agent, or process compatibility of a new component or device.

For more information on cleaning, refer to IPC-CH-65*, Guidelines for Cleaning of Printed Boards and Assemblies*.

Section 9 Inspection

The Hirose IT3/IT5 receptacles' 6mm or 7mm heights allow for high-resolution, low-noise x-ray imaging. Whether a mounting or mating receptacle, and regardless of the stacking height of the final assembly, the images remain consistent. No special modifications to the image analysis routine to compensate for different connector heights are needed.

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

9.1 Sampling Frequency

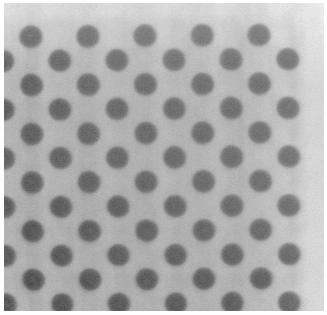
100% of the components should be inspected for the following defect modes:

- Solder Bridges
- Cold Solder Joints
- Open Solder Joints
- Cracks
- Head-In- Pillow*
- Excessive Voids

** The **appearance** of **head-in-pillow** defects in **ball-on-pin** BGA constructions **differs** slightly from their typical appearance in **ball-on-pad** constructions. Refer to section 11, Tips for SMT Assembly for more information on head-in-pillow.*

9.2 X-Ray Laminography

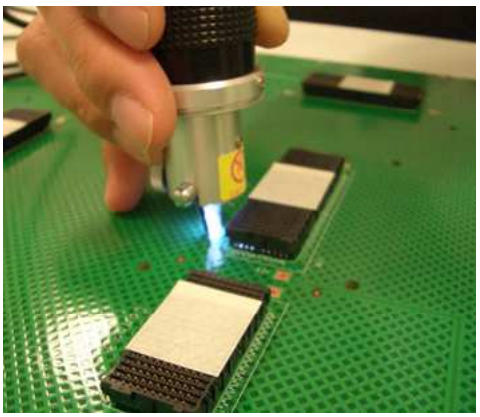
As part of the production process, 3-dimensional x-ray laminography should be used to inspect the assembly after reflow processing.

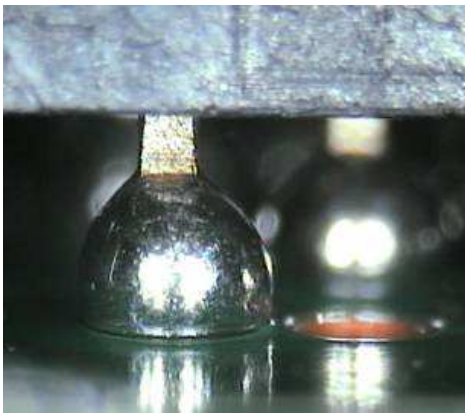


If the automatic inspection system reports potential soldering defects, **the defects should be verified** by visual inspection and/or 2-dimensional transmission x-ray inspection with oblique angle views before beginning the rework procedure.

9.3 Visual Inspection

If the suspected defect is near the outside edge of the receptacle, it may be visible using optical devices. Visual inspection tools include devices ranging from hand-held mirrors to digital endoscopes. If the suspected defect is on an interior ball, it may require 2-dimensional x-ray for verification.

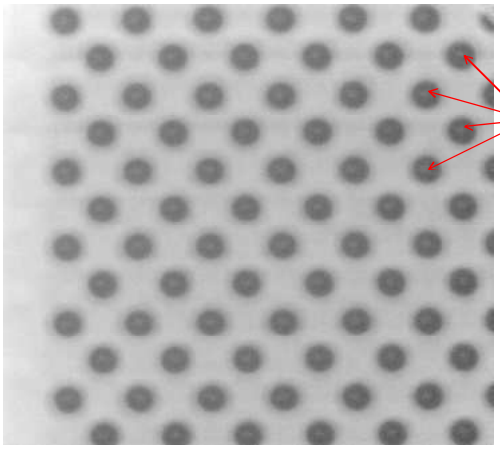


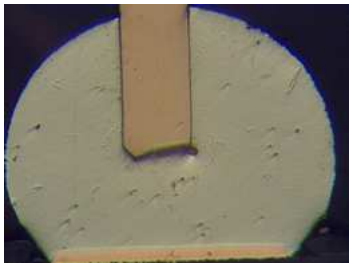


9.4 X-Ray Analysis and Voiding

Levels of acceptability for voiding depend on the final application of the end product. For general guidelines regarding voiding, see IPC-7095, *Design and Assembly Process Implementation for BGA's*.

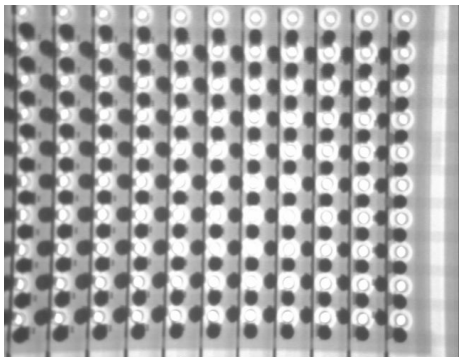
Note that the connector design is **ball-on-pin** style. If pins appear as voids during x-ray analysis, parameters should be adjusted to eliminate the false detection of voids.

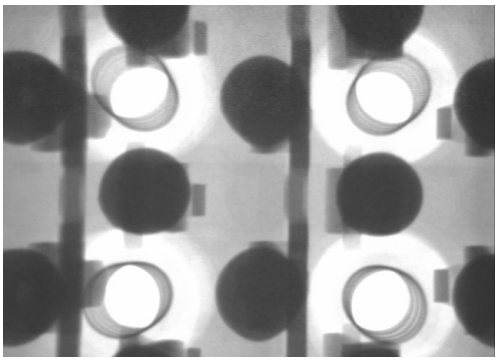




9.5 Transmission 2-Dimensional X-Ray

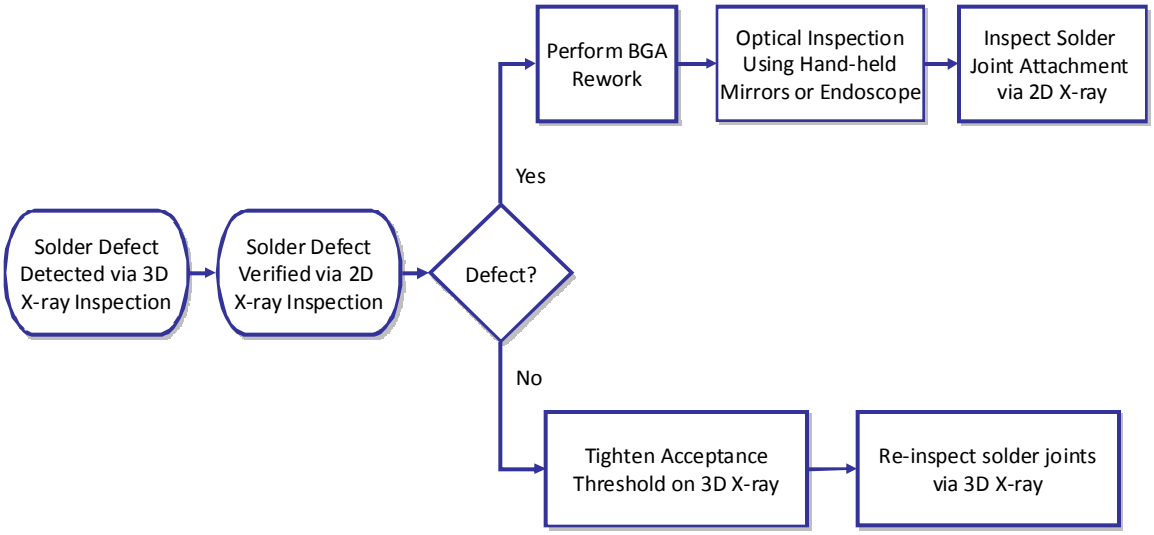
A 2-dimensional x-ray system can be used to verify certain defects that are indicated by the 3-dimensional inspection, particularly if the interconnections are viewed at oblique angles.





9.6 Automatic Inspection Optimization

See flowchart below for optimizing inspection routines:



Section 10 Rework

*Rework processing of the **IT3/IT5** connector receptacle is similar to that of other high-value BGA devices. Thermal profiles should be developed specifically for the receptacle, best available practices should be used for site redressing, and receptacles that are removed from PWB's should be discarded and not reused.*

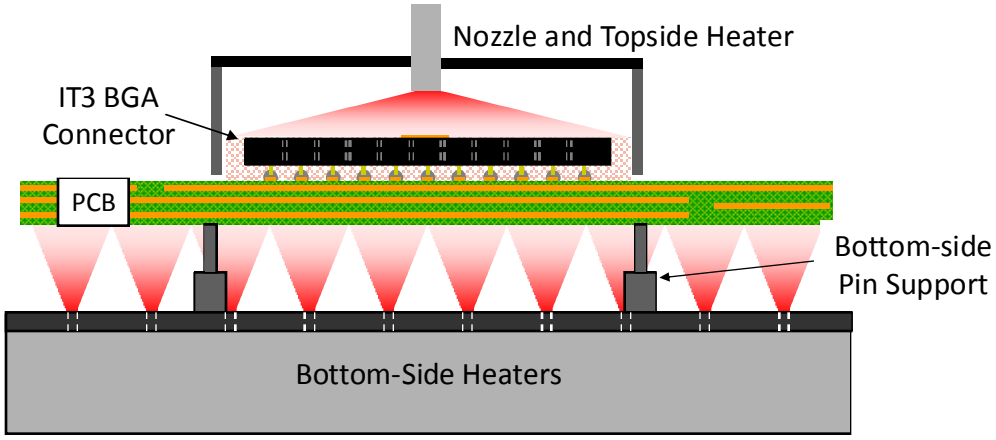
Rework on devices peripheral to an IT3/IT5 interposer may physically damage and melt the outer walls of the interposer. Please remove the interposer for such rework. Interposer removal procedure is on page 73.

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

10.1 Equipment and Supplies

- **Forced Hot Air BGA Repair Station with bottom side heaters, properly sized (custom) nozzle, vacuum pickup/low placement force capability, and split vision alignment system**
- **Adhesive polyimide (tape) discs 3/8" or 1 cm diameter**
- **Liquid flux**
- **Non-contact, vacuum redressing tools**
- **Rework stencil, mini-squeegee, and solder paste**
- **Hand-held inspection device(s)**

10.2 Profiling

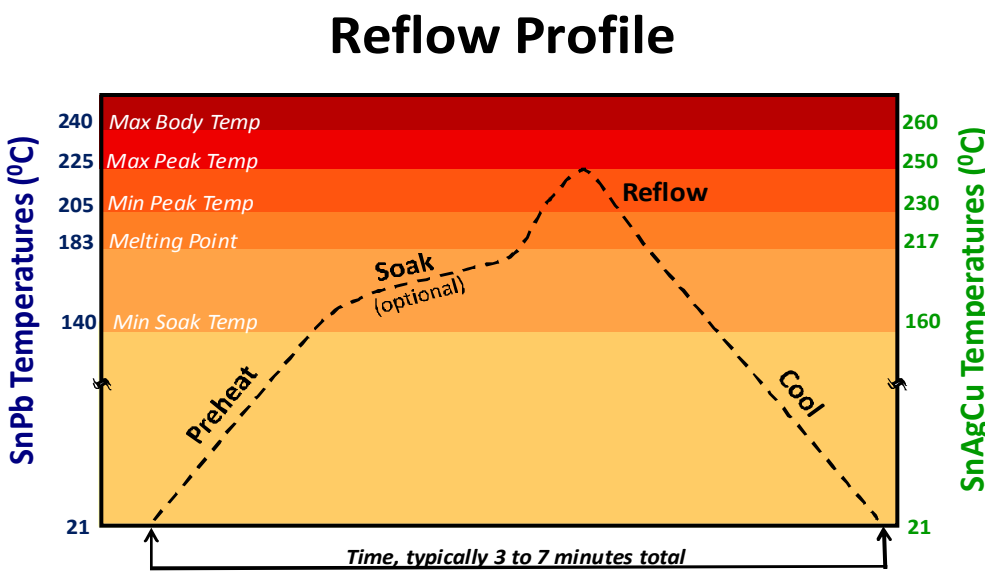


10.3 Reflow Profile Considerations

The ***IT3/IT5*** receptacle has an open body design that allows easy air circulation, but **bottom side preheat must be used to prevent PWB damage**. Removal and reattachment profiles should follow same guidelines as mass reflow process:

Parameters	Tin-Lead	Lead-Free	Comment
Preheat Ramp Rate	2 - 3°C/sec	2 - 3°C/sec	Other components may limit ramp rate to 2°C/sec
Soak Time	0 - 120 sec	0 - 120 sec	Soak requirements determined by board design, oven capability, and paste activation requirements
Soak Temperature	140 - 180°C	160 - 215°C	Caution - "oversoaking" may exhaust flux and affect soldering
Peak Reflow Temperature	205 - 225°C	230 - 250°C	Cooler peak temperatures may require longer TAL's
Time Above Liquidus (TAL)	30 - 90 sec	45 - 120 sec	Shorter TAL's may require higher peak temperatures
Cooling Rate	>6°C/sec	>6°C/sec	Faster cooling rates produce finer grain structures and smoother joint appearances
Maximum Package Body Temperature (T)	240°C	260°C	Open body design allows for low delta T between package and solder joint
Maximum Delta T between Body and PWB at Liquidus	10°C	10°C	Standard practice is easy to achieve with open body design
Package Body Exposure Limit at Maximum Temperature	5 sec	5 sec	Adjust profile if maximum exposure limit is approached or exceeded

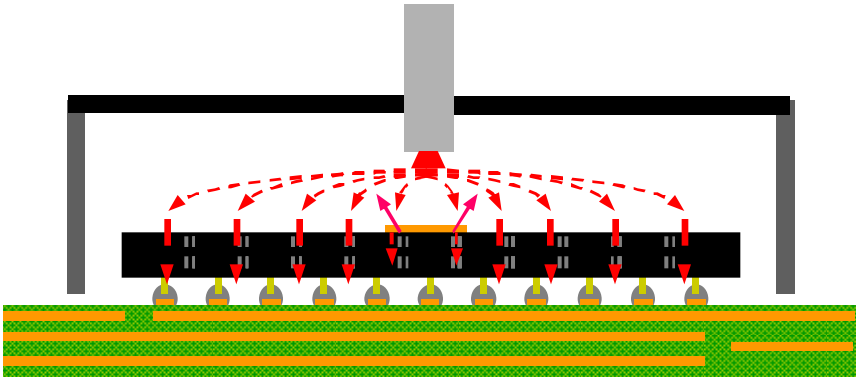
10.4 Suggested Thermal Profile Ranges



10.5 Special Profiling Considerations

Important profiling techniques for open-body package style:

1) **Place pick and place tape, polyimide disc, or metal cap on connector before profiling!** The open-body design of the connector aids in heat transfer, but the tape or cap used for placing the receptacle will block some of the heat flow under the area where it is placed, as shown below:



Joints under tape or cap get less airflow and stay cooler

The solder joints whose airflow is obscured by tape or cap will likely be the coolest on the connector. Care should be taken during the profiling process to verify adequate thermal exposure in these areas. **The tape or cap should not be peeled off the receptacle prior to reflowing the solder joints.**

2) **Component body should not be cooled during reflow heating**. Some convection rework systems offer the option of component body cooling during the rework process. This should **not** be used on the ***IT3/IT5*** connector. Its open body design allows good heat transfer and low delta T's between solder joints and component body. **Use of body cooling system may interfere with hot air flow and cause unnecessary temperature differentials.**

10.6 Receptacle Removal

Place polyimide disc or metal cap on center of connector where vacuum nozzle will pick up component. Using profiles generated with these techniques, remove receptacle with rework station. **Discard removed receptacle. Replace with new receptacle. Do not attempt to reball and/or reuse receptacle.**

10.7 Site Redressing

When the receptacle is removed from the PWB, some **excess solder will remain** on the pads. The remaining solder **must be thoroughly cleaned** off before new solder paste can be added.

Excess solder should only be removed with non-contact, vacuum solder removal systems. Some automated BGA workstations have solder removal systems built into them. These are generally the safest methods of removing excess solder.

Manual vacuum solder removal systems are also effective, but require more operator care to ensure non-contact with the board and limit damage due to scraping with the nozzle or localized overheating. If manual vacuum solder removal is necessary, the PWB should be preheated to 150°C if tin-lead solder is used and 180°C if lead-free solder is used,

Soldering irons and wick should not be used to remove excess solder. This method provides the highest risk of lifted pads, torn traces, or damaged solder mask. If wick must be used to remove the excess solder, see section titled *Solder Wicking Guidelines* on next page.

Vacuum Solder Removal Guidelines

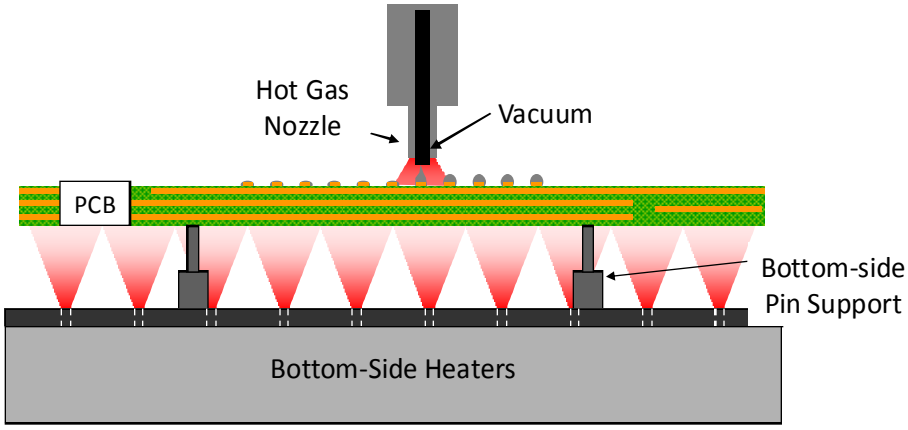
Liquid flux should be dispensed onto the PWB to help the solder flow during the redressing operation, especially if the redressing process uses air. Liquid flux may be optional if the redressing operation uses nitrogen.

Preheat the PWBs to **150°C for SnPb**, or **180°C for SAC alloys**.

Typical **nozzle temperatures** for removing excess solder are **280°C for SnPb** and **340°C for SAC**.

Clearance between the nozzle and the PWB should be approximately **125 um**.

Please **consult equipment manufacturer** for specific instructions on site redressing process.



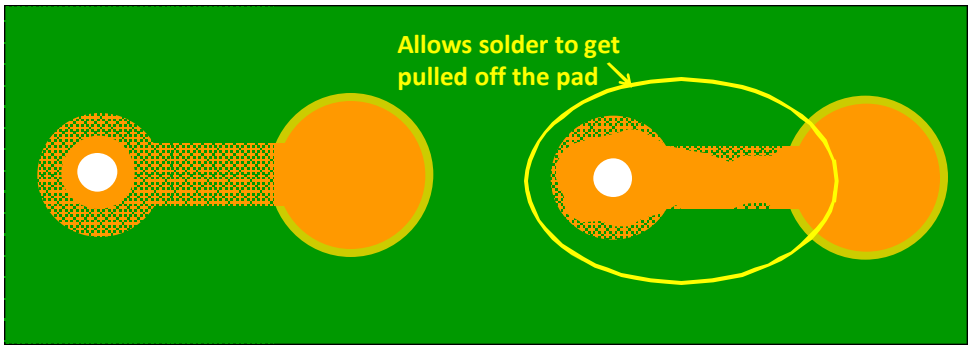
Solder Wicking Guidelines

If **solder wick** is the only available method of removing the excess solder, the operation should be performed only by an experienced, certified rework specialist. A **fluxless braid** (wick) should be used in conjunction with liquid flux. The braid should be **no more than 2-3 balls wide**, and used with a blade-tip soldering iron. **Liquid flux must be used** in this process. *If the solder braid contains flux, its compatibility with the liquid flux should be verified before the two chemistries are used together.*

Precautions: **Some solder should remain on the pads** to help prevent exposure of the intermetallic compound (IMC) layer to the atmosphere, which can allow it to oxidize. An oxidized IMC layer is less solderable than a properly redressed pad and could result in an open interconnection after the repair.

Inspection

After solder removal, the site should be thoroughly cleaned and dried before continuing. At this time, the **pads and solder mask should be inspected under a microscope** to ensure they have not been damaged and are ready to receive a new component. It is very important to check for solder mask damage between a pad and its corresponding via, as damage to the solder mask dam can cause open or insufficient solder joints.



Non-Damaged Soldermask Dam **Damaged Soldermask Dam**

10.8 Solder Replenishment

A small metal or flexible polymer stencil should be used to **apply solder paste to the circuit board.** The stencil should use the same aperture design as the original SMT stencil; 0.54mm circles are preferred. ***A good quality solder paste print is absolutely necessary for successful rework.*** Metal stencils should be positioned over the pads and taped to the PWB along the edges to hold them in place. Most polymer stencils have residue-free adhesives that will hold them to the board during the printing process. At this step, proper alignment between the stencil apertures and the pads is critical.

Some BGA rework procedures include printing paste on the balls of the BGA. Due to the compliant nature of the ball-on-pin design of the ***IT3/IT5*** connector, **printing paste directly on the BGA balls is not recommended.**

Precautions: **Solder MUST be applied to the pads during the rework process. Use of flux alone has been shown to result in reduced reliability of the assembly.**

10.9 Receptacle Placement

Receptacle should be aligned using the optics on the automated rework system.

Placement force should be minimal to avoid disturbing the solder deposit.

Metal cap should be used on 300 position receptacles to maintain stiffness during rework.

10.10 Reflow Soldering

Receptacle should be reflowed using profile developed specifically for this component/assembly. A **nitrogen atmosphere** is typically **preferred during** the **reflow** stage.

10.11 Inspection

Receptacle should be **visually inspected first**, using hand-held mirrors or a digital endoscope.

Reworked solder joints should have same appearance as non-reworked solder joints.

If no defects are noted during visual inspection, it should then be evaluated by 2-dimensional x-ray. If no defects are noted under the 2-D x-ray inspection, rework can be considered successful.

10.12 Rework Solder Paste Selection

Precautions: **The type of solder paste used in rework should be selected carefully. The alloy (tin-lead or lead-free) should be verified before beginning rework procedure.**

Additionally, mixing flux chemistries can cause reliability problems. See chart on next page for flux chemistry selection suggestions.

Oringial Paste Type	Cleaned?	Rework Paste Type	Clean After Rework?
No Clean	No	No Clean	No
No Clean	Yes	No Clean	Yes
Water Soluble	No	Water Soluble	Yes
Water Soluble	Yes	No Clean	No

**If water soluble paste is used and the assembly has not yet been cleaned when the rework is performed, compatible water soluble solder paste may be used for the repair. If the assembly has already been washed and sensitive components have been added, no-clean paste may be required.*

Section 11 Tips for SMT Assembly

The SMT assembly process has many sources of variation, introduced by the equipment, the components, the chemistries and the environment. Most common defects can have many possible root causes located anywhere throughout the process. Please note that:

1) *Before making changes to the assembly process, it is very important to verify the root cause(s) of defects.*
2) *After making changes to the assembly process, it is very important to follow some assemblies through the remainder of their production to make sure the changes do not create other problems downstream.*

Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

11.1 Stencil Printing

Of all the steps involved in and SMT assembly process, stencil printing has the largest influence on overall yields. Therefore, getting a good quality stencil print plays a key role in a high quality process. If print quality is not satisfactory, corrective action should be taken immediately.

System check: The **first step** in investigating print quality problems is usually to **check the setup** of the printing system. A typical setup check would include:

1. **Observe the solder paste**. Ensure the proper amount is on the stencil – the diameter of the bead should be approximately 1.5cm. Watch it during the print stroke. It should roll over the surface of the stencil, and should release cleanly from the squeegee blade when the blade lifts away from the stencil. If the paste does not show this desired behavior, remove it from the stencil and replace it with fresh solder paste.
2. **Remove the stencil** from the printer and **check it for damage**, such as broken or distorted webs between the apertures in fine pitch areas, or dents or deformations anywhere in the foil**. Check the apertures** for debris or residual solder paste. **Check the fiducials** for wear or solder paste smears. **Check the stencil's mounting mesh** for rips or tears.
3. While the stencil is out of the printer, shuttle the board into printing position. **Check its underboard support** by gently pressing on the PWB with a finger. **Check its lateral support** by trying to move it in the X and Y directions**. Always wear gloves** when handling bare PWBs. If vacuum tooling is used, check it for leaks.
4. **Reinstall the stencil** and monitor its automatic alignment process. With the stencil in print position (contacting the PWB), visually **confirm that the apertures and the pads are properly aligned**. Tap gently on the stencil foil to confirm solid contact between the stencil and the PWB. **There should be no gaps between the stencil and the PWB.**

5. **Inspect the squeegee blades** for the proper **angle** (usually 45 degrees), damage or **nicks on the blade's edge**, or **dried paste** stuck to the blades or holders. **Check pressure balance** on blades.

If no issues are found during review of the printer's setup, please refer to the following specific guides for additional troubleshooting assistance.

Solder Paste Bridges: Are often referred to as **"wet bridges"** and occur when the normal gaps between two or more adjacent pads is bridged by stray solder paste.

Solder Bridges or Strings	Possible Cause	Suggested Action
	Poor gasketing	See section on gasketing below
	Residual paste from previous print(s)	Check stencil cleaning parameters, increase cleaning frequency
	Separation Speed (too fast or too slow)	Increase or decrease separation speed (individual solder pastes have unique optimums)
	Squeegee pressure too high	Decrease pressure. It should be just enough to get a clean wipe across top surface of stencil
	Too much paste on stencil	Check paste bead on stencil – 1–1.5cm diameter is typical
	Paste is too warm	Check paste temperature and consult paste manufacturer's recommendations

Print Definition: is considered good if all **deposits are** of **uniform shape and size**, **with well defined edges** and **flat** or slightly dome-shaped **tops.**

Poor Print Definition ("Dog Ears" or Peaks)	Possible Cause	Suggested Action
	Poor gasketing	See section on gasketing below
	Separation Speed (too fast or too slow)	Increase or decrease separation speed (individual solder pastes have unique optimums)
	Residual paste from previous print(s)	Check stencil cleaning parameters, increase cleaning frequency
	Misalignment	See section on alignment below
	Squeegee pressure too high or too low	Adjust pressure. It should be just enough to get a clean wipe across top surface of stencil.
	Paste is too warm	Check paste temperature and consult paste manufacturer's recommendations

Insufficient Solder Volumes: can have two different causes: **stencil apertures not filling, or not releasing**. If solder deposit volumes are insufficient, **observe stencil apertures** to determine if problem is due to solder paste not filling aperture or not releasing from aperture. If, after printing, apertures are relatively clean, refer to chart titled "Apertures Not Filling." If apertures show excessive residual solder paste, refer to chart titled "Apertures Not Releasing."

Apertures Not Filling

Insufficient Solder Volumes (Apertures Not Filling)	Possible Cause	Suggested Action
	Interruption (pause) in printing raised viscosity of paste	Knead paste 4-10 strokes before resuming printing. Thoroughly clean board used for kneading paste.
	Squeegee speed too high or too low	Check print speed
	Squeegee pressure too low	Adjust pressure. It should be just enough to get a clean wipe across top surface of stencil.
	Not enough paste on stencil	Check paste bead on stencil – 1–1.5cm diameter is typical
	Paste is too cold	Check paste temperature and consult paste manufacturer's recommendations.
	Paste sticking to squeegee	Check paste bead on stencil. 1–1.5cm diameter is typical. Check paste temperature
	Squeegee damaged or worn	Inspect blades and replace if necessary

Apertures Not Releasing

Insufficient Solder Volumes (Apertures Not Releasing)	Possible Cause	Suggested Action
	Interruption (pause) in printing raised viscosity of paste	Knead paste 4-10 strokes before resuming printing. Thoroughly clean board used for kneading paste.
	Residual paste building up in apertures	Check stencil cleaning parameters, increase frequency
	Paste is too cold	Check paste temperature and consult paste manufacturer's recommendations
	Squeegee pressure too low	Adjust pressure. It should be just enough to get a clean wipe across top surface of stencil.

Bad Gasketing: Poor **gasketing** is the root cause of many printing problems. If the **edges of the stencil aperture** cannot **seal against the PWB pad**, solder paste gets squeezed through the resulting gap and can cause a variety of defects.

	Possible Cause	Suggested Action
Gasketing Problems	Poor Board Support	Check and improve PWB support
	Alignment	See section on Alignment below
	Solder mask higher than pads	Check solder mask height and compare to specification.
	Stencil apertures are larger than PWB pads	Measure apertures and pads and compare to specification.
	Hot Air Solder Level (HASL) Finish creates uneven printing surface	Consider more planar, non-HASL finishes. Consult with PWB fabricator on improving doming effect of hot air leveling.
	Labels, inks, or other surface features create a gap between stencil and PWB	If defects occur near surface features, consider changing placement of those features, or half-etch stencil from underside to allow clearance for feature.

Bad Alignment: An **alignment error of up to 50 microns is typical** and usually acceptable, depending on tolerances in other parts of the assembly process. If alignment of paste deposits and pads cannot be maintained at less than 50 microns, corrective action should be taken.

	Possible Cause	Suggested Action
Alignment Problems	Poor Board Support	Check and improve PWB support
	Printer alignment error	Check printer fiducial reading errors
	Stencil mesh torn or tension too loose	Check stencil's mobility in frame. Look for movement at beginning of print stroke.
	PWB or stencil positional error	Check corner-to-corner alignment of apertures and pads. If they cannot be aligned, stencil should be scaled to compensate for error.

11.2 Pick and Place

Pick and Place problems are often the easiest to troubleshoot, as the root cause of most potential causes are within the pick and place machine itself.

Vision Errors: can cause the pick and place system to **reject or misplace** good components.

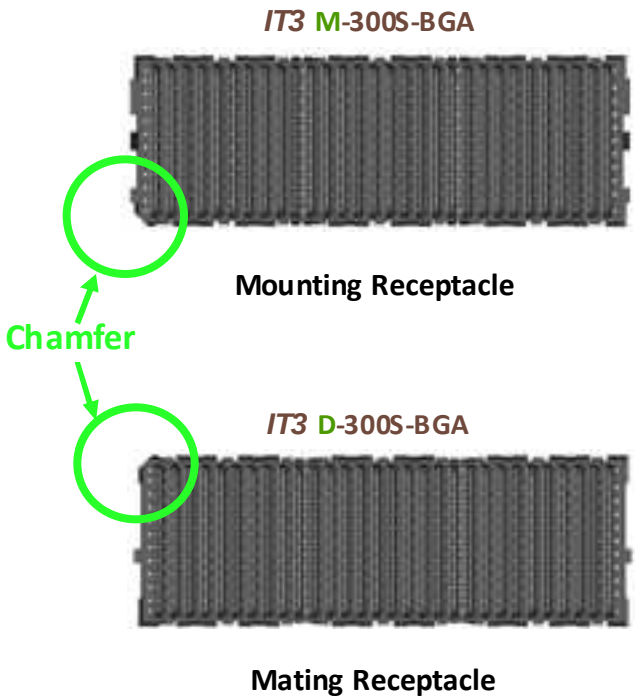
	Possible Cause	Suggested Action
Vision Rejects	Lighting	Close any open covers on machine. Check lighting and adjust as necessary to provide proper illumination of the balls.
	Inspection Criteria	Review inspection routines - search windows, lighting thresholds, etc.
	Wrong Orientation	Check component orientation in trays, tray orientation in feeders
	Feeders malfunctioning	Check feeder operation, alignment, advance

Misplacement: **BGA device**s have **self-centering properties** in the reflow process. It is suggested that the ***IT3/IT5*** **connectors should be placed no more than 0.1mm off-pad.** If machine is placing connector receptacle more than 0.1mm off-pad, investigate the root cause.

	Possible Cause	Suggested Action
Misplacement	Poor vision capture miscalculated centroid	Check vision inspection system. Make sure to inspect all balls, not just corner balls.
	Wrong Orientation	See section on orientation below
	Component shifting on nozzle	Check nozzle for obstructions, make sure nozzle size is appropriate for component weight
	Programming Error	Check programmed coordinates

Wrong Orientation: If **component orientation** is 90 or 270 degrees off, check pick and place program first. If orientation is 180 degrees off, review the following steps to assure the right part is loaded properly before modifying the pick and place program.

	Possible Cause	Suggested Action
Wrong Orientation	Tray placed in feeder wrong	Check tray's position in feeder
	Component placed in tray wrong	Check components' positions in trays
	Mounting and mating receptacles mixed up	Check to see which corner chamfer is on; compare with drawings below
	Rotation in program off by 180 degrees	Check programmed theta angle



11.3 Reflow

Defects that appear after the reflow process may or may not be attributed to the process itself. Many factors can affect the formation of defects during reflow soldering. If solder defects are found, all sources should be investigated before any modifications are made to the reflow profile.

Solder Bridges: **Solder shorts**, or **bridges**, occur when two or more adjacent conductors are connected by solder.

	Possible Cause	Suggested Action
Solder Shorts	Poor solder paste print	Check paste print quality
	Placement pressure is too high and deforms solder paste deposit	Check placement pressure and depth of placement
	Reflow profile taking too long to reach liquidus temperature, causing hot slump and/or and exhausting the flux	Shorten time to reach liquidus by ramping faster or limiting soak time
	Component shifting during transport	Check conveyor for rapid starts and stops, and any component movement during transport or placement

Opens: **Solder opens** occur when the connection between a ball and a pad is not completed.

	Possible Cause	Suggested Action
Solder Opens	Poor solder paste print	Check paste print quality
	Device warpage lifting balls out of paste	Shorten time to reach liquidus by ramping faster or limiting soak time
	Reflow profile not reaching high enough peak temperature, resulting in cold joints	Increase peak temperature and/or time above liquidus

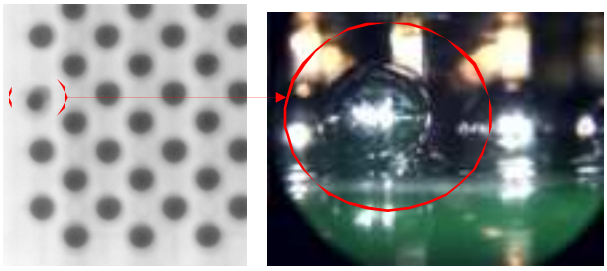
Head-In-Pillow (HIP): defects generally occur when:

- The solder paste deposit on the board melts and fuses together in the reflow oven
- The solder sphere on the BGA also melts in the reflow oven
- The sphere material and solder deposit **do not** fuse together to become one contiguous mass.

Typically, the reason they do not fuse together is because one or both of the molten masses are covered with a thin but tough film of oxide that prevents them from mingling with each other. This oxide film is often formed in the reflow process by one or more contributing factors.

Note that the HIP defect mode has a slightly different appearance on ball-on-pin style BGA components than it does on ball-on-pad style components.

Head-In-Pillow Defect,
Ball-on-pin style BGA



Notice how the ball gets offset from the solder and pad under HIP condition

	Possible Cause	Suggested Action
Head-In-Pillow	Reflow profile taking too long to reach liquidus temperature, exhausting the flux	Shorten time to reach liquidus by ramping faster or limiting soak time
	Component body warpage lifting balls out of paste	Shorten time to reach liquidus by ramping faster or limiting soak time
	Surface area:volume ratio of paste deposit enables flux exhaustion	Increase diameter of stencil aperture to 0.6mm.

Voids: **Are small pockets of gas** that get trapped in solder joints. There are many different types of voids and possible causes, but the only type that can be addressed through reflow profiling are produced by solder paste and referred to as **process voids.**

	Possible Cause	Suggested Action
Excessive Voiding	Process voids, volatiles in solder paste did not have enough time/temp to outgas	Increase soak time or temperature; consult solder paste manufacturer's guidelines
	Voids in center of each ball	Verify that the lighter areas in the centers are pins and adjust the inspection routine (see section on false calls)
	Microvoids, at the PWB interface	Contact PWB supplier

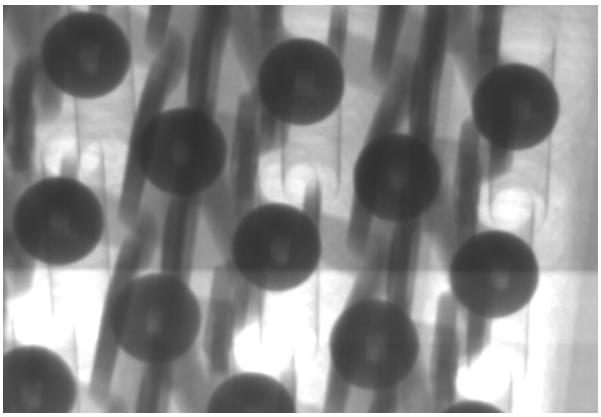
Solder in Vias: If **solder** is detected **in a via**, it has likely been taken from a neighboring joint, leaving that **joint with insufficient solder**.

	Possible Cause	Suggested Action
Solder in Via	Poor solder paste print	Check paste print quality
	Damaged solder mask dam	Inspect mask between pads and vias for flaking or other signs of poor adhesion

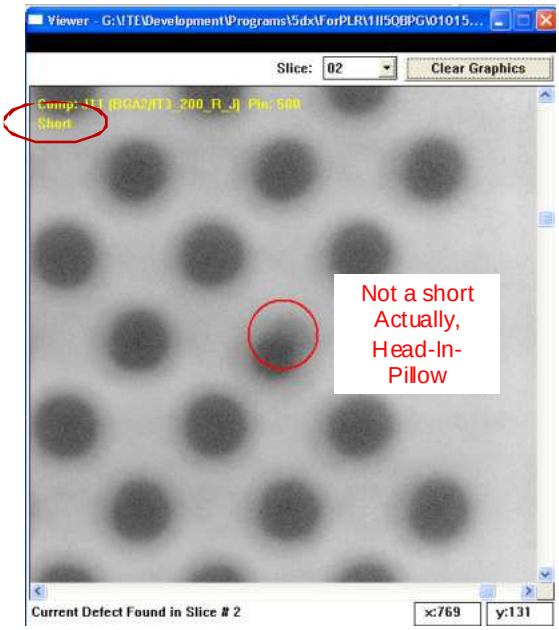
11.4 X-Ray Inspection

False Calls: Automated X-ray Inspection (AXI) *is an effective way to screen for defects, but depending on the specific equipment of programs, can sometimes return false failures.*

	Failure Mode	Suggested Action
AXI False Calls	Voids in center of every ball	At certain x-ray settings, the pins can appear as lightened areas in the center of each ball (see example below). Modify inspection routine.
	Shorts detected when they do not exist	Check for Head-In-Pillow defect mode (see example below)



If the pins are being detected as voids during x-ray inspection, modify the inspection routine to eliminate the false failure



HIP defects may be detected as shorts, because the condition forces the ball's image outside of the normally recognized ball area of the x-ray program. If images are captured that show offsets similar to the one above, check for head-in-pillow defect.

11.5 Rework

Most BGA rework issues have one of two root causes:

- Solder paste print quality
- Reflow profile

Paste Prints: Print-related defects include:

	Failure Mode	Suggested Action
Print Quality	Solder shorts	Solder shorts are often from solder paste smears on the bottom of the rework stencil. For best results: • Thoroughly clean and dry both sides of stencil and its apertures and before use • Check PWB pads for proper site redressing • Tape stencil to PWB to prevent slippage during printing • Fill apertures with 1-2 strokes of the squeegee blade. Do not go over stencil multiple times as it can force unwanted paste into areas between stencil and PWB • After printing, remove tape and pull stencil straight off of PWB. Do not use peeling motion to remove stencil. • Inspect print for sufficient volumes, good definition, and alignment with pads
	Opens	Opens are often the result of insufficient solder paste deposits. Following the printing process described above: • Make sure all apertures are completely filled before removing the tape from the stencil. • After removing the stencil from the PWB, check the apertures to make sure the paste released. If paste does not release well, check its temperature and condition. If it is too cold, warm the paste sample as per the manufacturer's directions. If it is dried out from atmospheric exposure, discard and replace it.

Precautions: **Print solder paste on PWB only**. Do not print paste on balls. The IT3/5 connector systems use a compliant ball-on-pin design. Printing on the balls may affect their alignment.

Always use solder paste for ***IT3/IT5*** rework. For maximum interconnection reliability, rework should be performed with solder paste, not flux alone.

Reflow: Defects related to reflow include:

	Failure Mode	Suggested Action
Reflow	Cold solder joints – only some portions of component affected	Verify profile. Check for proper nozzle on machine. Check position of PWB with respect to nozzle and bottom heater.
	Cold solder joints – entire component affected	Verify profile Verify solder alloy in paste (tin-lead or lead-free) Check location of trigger thermocouple
	Defects (bridges or opens) near the corners of the component	Shorten time to peak temperature of thermal profile. Keep delta T across component under 10 degrees.
	Package warpage	Shorten time to liquidus of thermal profile. Ensure parts are dry
	Poor solder paste fusion	Shorten time to liquidus of thermal profile. Use nitrogen as reflow medium
	Head-In-Pillow	Shorten time to liquidus of thermal profile Ensure balls are not contaminated or oxidized
	Nearby components affected	If secondary reflow or charring of flux residue occurs on nearby components, shield them from the heat/air flow of the rework nozzle.
	Solder in via	Check solder mask for damage

Do not re-use receptacles. If processing defects require rework, remove the ***IT3/IT5*** receptacle from the PWB and replace it with a new one.

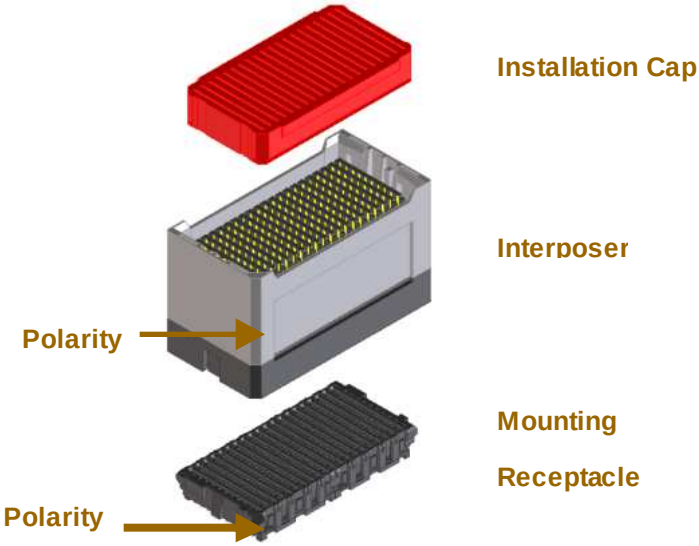
Section 12 System Level Assembly

*The Hirose **IT3/IT5** connector system's separation height is set by interposers that are installed at the system level, after the circuit assembly process is complete. The interposers can be installed by hand, with common hand tools, or with customized fixtures. For high assembly volumes, dedicated tooling is suggested to ensure quality and ease of operation.*

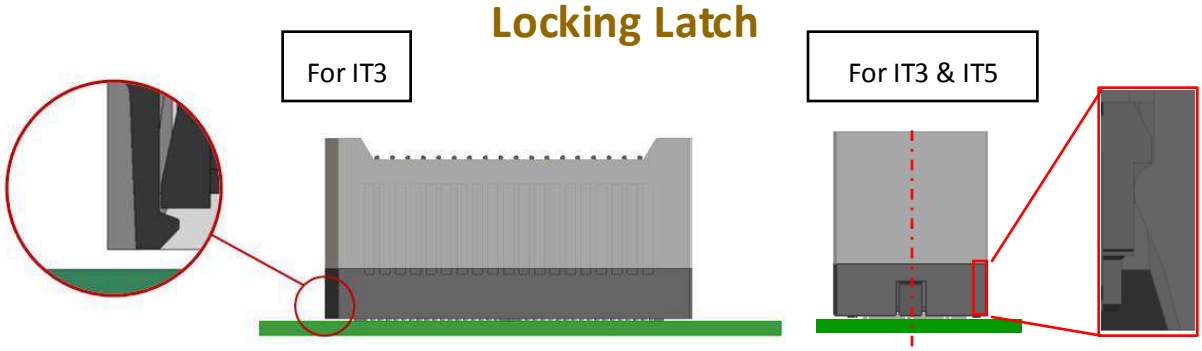
Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

12.1 Interposer Installation

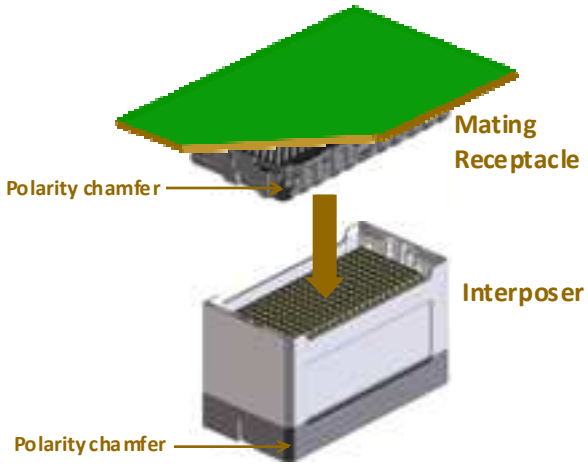
The interposer snaps on to the mounting receptacle as shown below:



The snap fit is achieved by a locking latch on each end of the interposer:

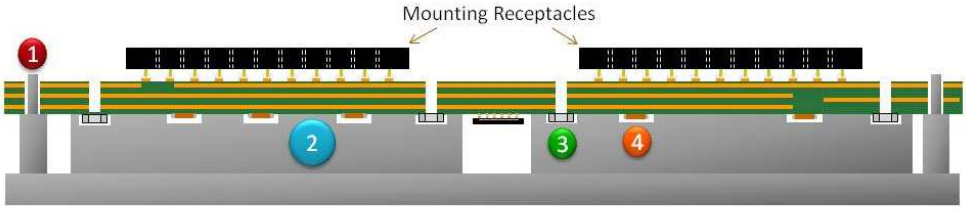


The spacers are installed (not shown) and the mating receptacle is aligned with the interposer and pressed on as shown below:



Precaution: It is **very important** to provide **good underside board support** when installing the interposers. A simple **tooling plate** can be fabricated to support the PWB and prevent it from flexing when the interposers are installed:

System Assembly Support Fixture



Key Features of Support Fixture:

1. Guide pins for PWB tooling holes align motherboard to support fixture *(shown 2 places)*
2. Support blocks directly under mounting receptacles prevent board from flexing during interposer and daughter card assembly* *(shown 2 places)*
3. Nests in blocks for spacer nuts hold the nuts in place while the spacer is tightened *(shown 4 places)*
4. Openings in block provide ample clearance for components *(shown 5 places)*

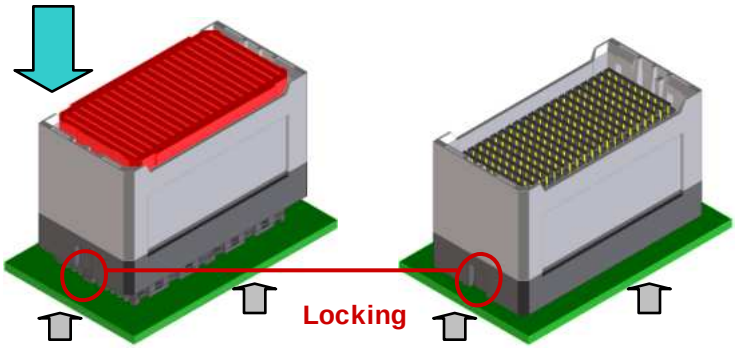
* For more information on PWB support and allowable deflections, reference IPC-JEDEC 9704, *Printed Wiring Board Strain Gage Test Guideline.*

12.2 Manual Assembly

Position interposer directly **over mounting receptacle**, aligning the polarity chamfers. When positioned properly, the interposer should slide easily onto the mounting receptacle. Place an installation cap. The side with horizontal grooves should face up. **Push vertically down** to engage the locking latches through an installation cap.

Manual Installation

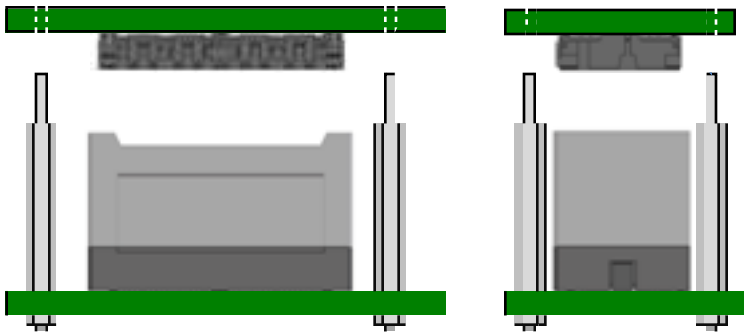
Press firmly on installation cap only,



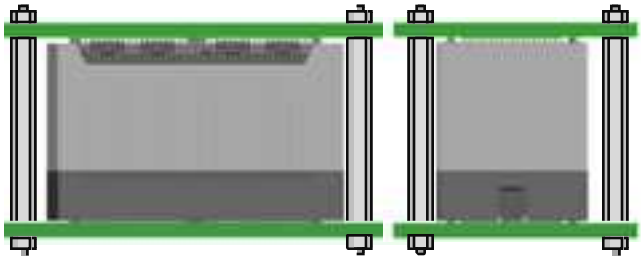
Always support PWB from underside to prevent flexing

After the interposer is mounted, install spacers onto motherboard (not shown).

To install mating receptacle, **align the spacer holes in the daughter card with the threads on the spacers.**

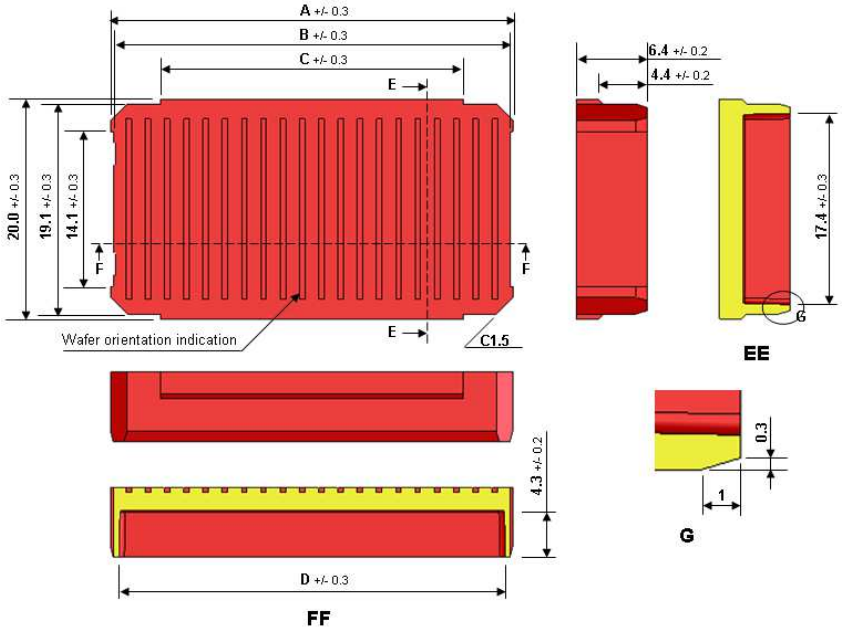


The spacers help align the mating receptacle with the interposer. If positioned correctly, the mating receptacle will slip down into the interposer. **Push directly down** on the assembly to lock the mating receptacle in place. Install nuts onto the spacer threads. Tighten nuts to specified torque.



12.3 Installation Cap

The installation cap is recommended to ensure quality and ease of operation during interposer installation. Caps are reusable, but should be discarded if they show signs of damage or wear. They should be stored properly when not in use and kept clean of dust and debris.



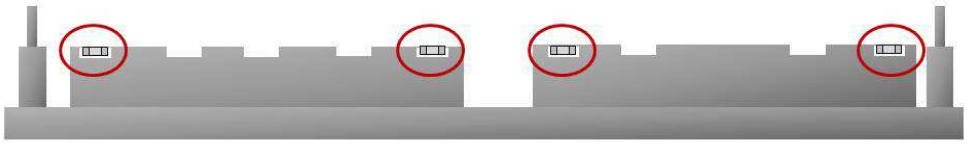
	Contact Position		
	100 POS.	200 POS.	300 POS.
A	19.0	36.5	54.0
B	18.5	36.0	53.5
C	10.0	27.5	45.0
D	17.6	35.1	52.6

12.4 Assembly Fixturing

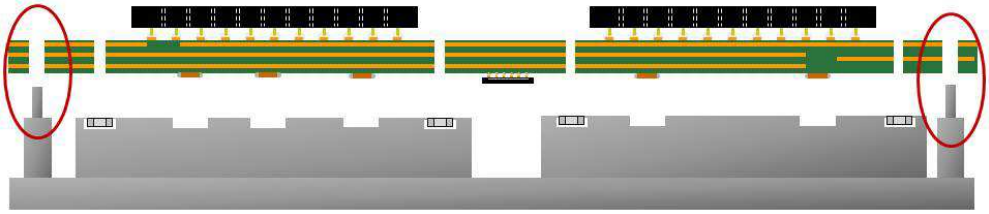
For most consistent results, and to improve ease of operation, **a simple fixturing system is suggested:**

Board-to-Board Assembly Fixturing

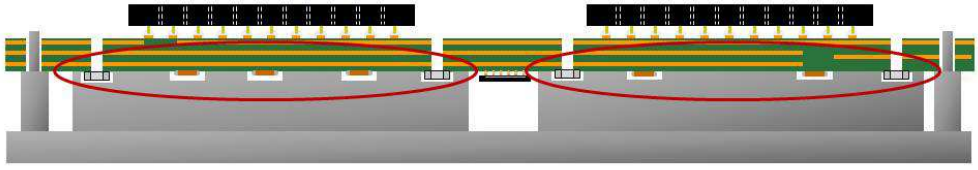
Insert spacer nuts into nests in assembly fixture.



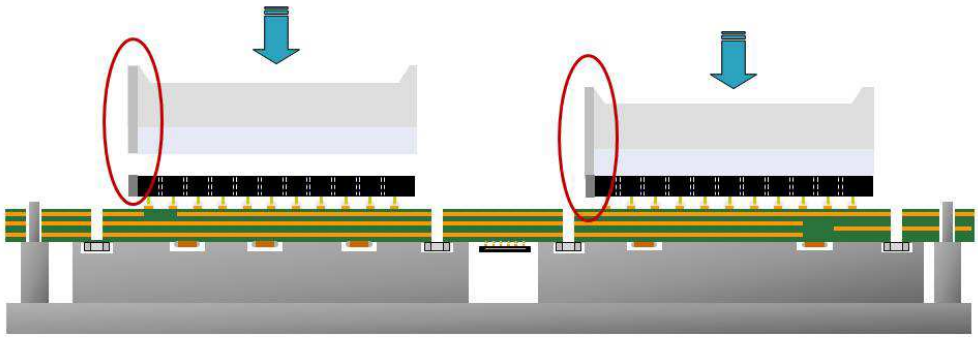
Place motherboard on fixture.
Align guide pins with PWB tooling holes.



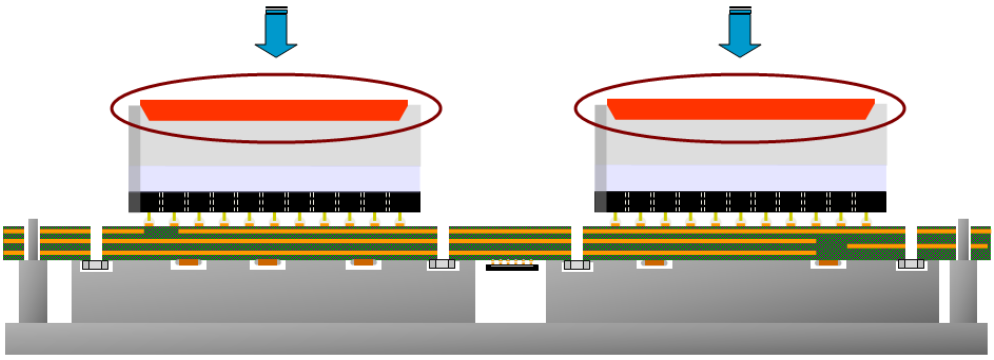
Assure motherboard sits flat on fixture.



Place interposers on top of mounting receptacles.
Align interposers and receptacles by their chamfered corners.

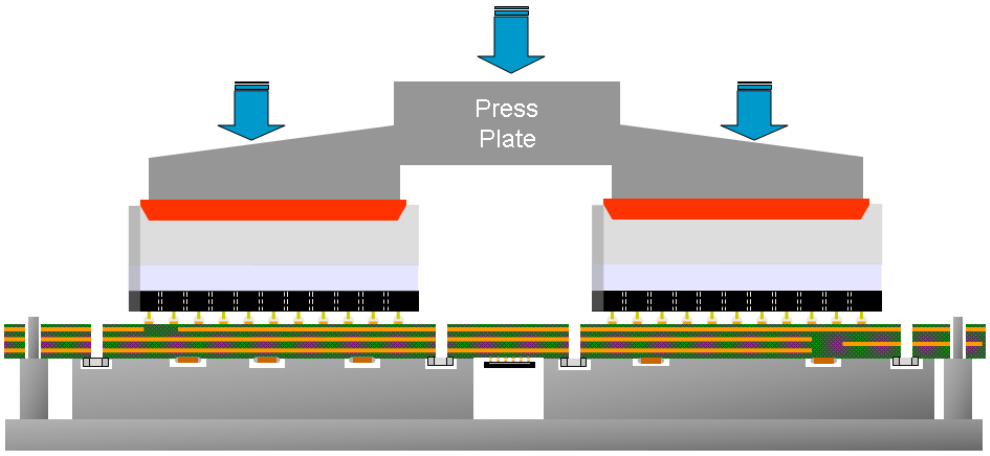


Insert a reusable plastic interposer installation cap into each interposer.



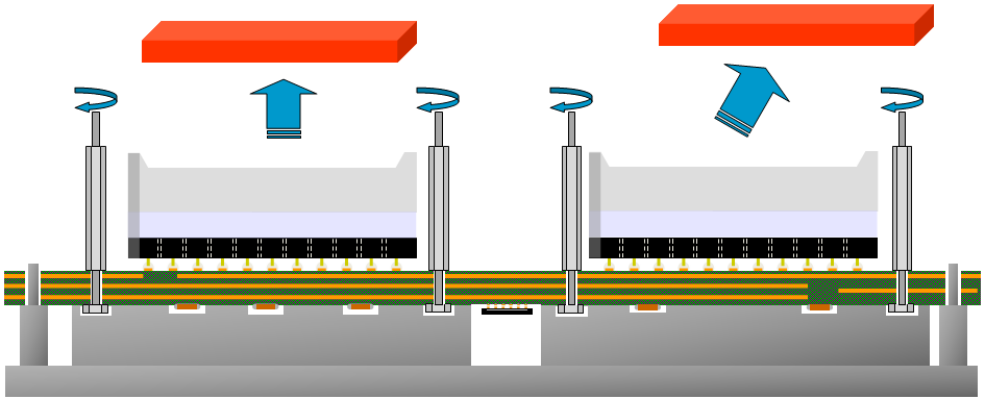
Caps should be clean, free of oil, dust or debris.
If cap is dirty or worn, replace with new cap.

Use top press plate to push directly down on interposers and lock them into place.
Press force should not exceed 500N for 300 pos, 400N for 200 pos, and 300N for 100 pos.
If a mechanical press is used, force must be measureable and controlled.



Example:
One 200-signal connector: do not exceed 400N insertion force
Two 200-signal connectors (400 signals): do not exceed 800N insertion force.

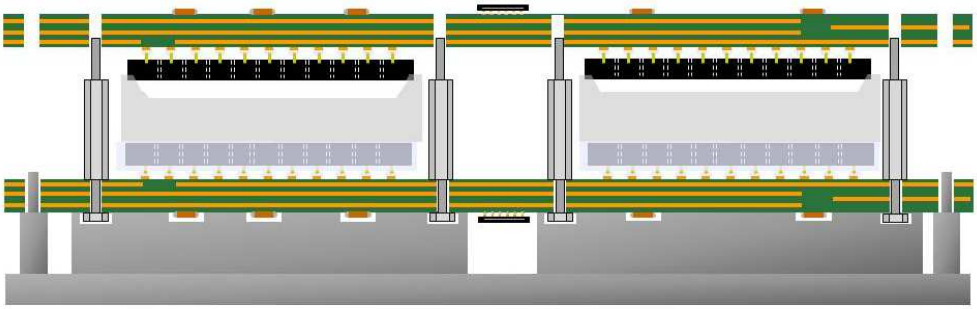
Install spacers. Tighten to specified torque.
Remove protective interposer installation caps. Save caps for reuse.



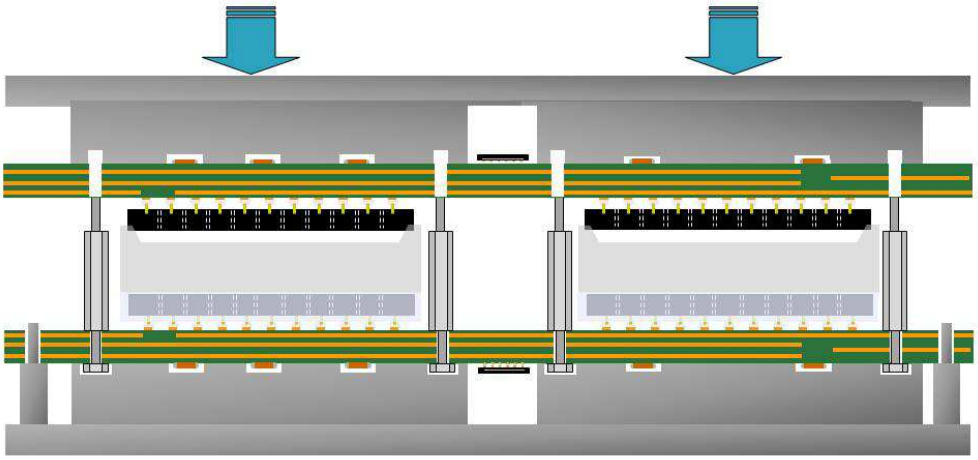
Leave caps on during spacer installation process to protect wafers from dropped fasteners or tools.

Precaution: It is very important to tighten nuts to torque specified by OEM. Failure to follow designers' recommendations can result in system damage or decreased reliability.

Position daughtercard over motherboard. Use spacer studs for initial alignments.
When mating receptacles are aligned with interposers, board will drop into place.

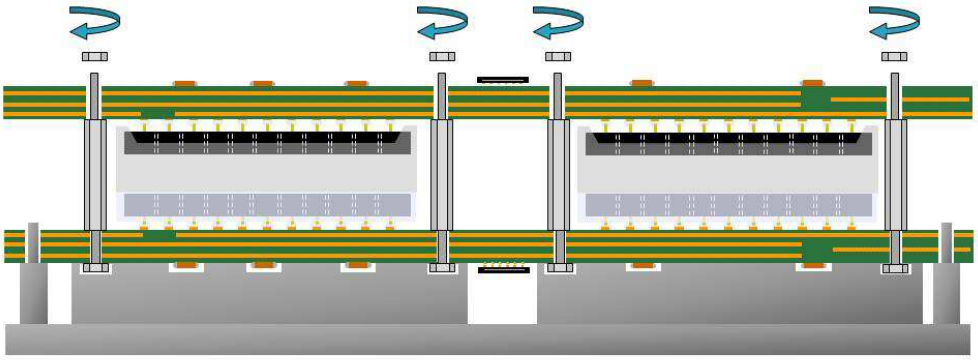


Use press plate to lock mating receptacles into interposers.
Press force should not exceed 500N for 300 pos, 400N for 200 pos, and 300N for 100 pos.

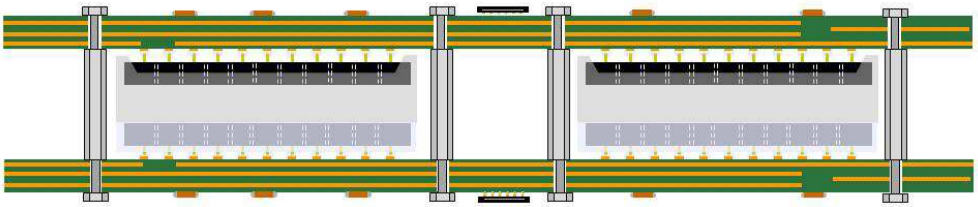


Example:
One 200-signal connector: do not exceed 400N insertion force
Two 200-signal connectors (400 signals): do not exceed 800N insertion force.

Install nuts on spacer studs. Tighten to specified torque.



Assembly mating process is now complete.



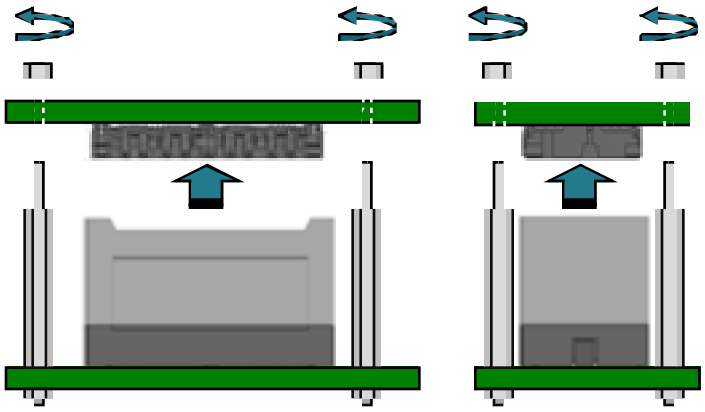
Section 13 System Level Disassembly

*The Hirose **IT3/IT5** three-piece connector system can be disassembled if a mother board or daughter card requires replacement. Both the mating receptacle and the interposer are removable. When removing a card or a connector component, the circuit boards should be handled with great care to prevent damage to them.* ***Disassembly should only be performed in authorized service centers by trained personnel using appropriate tools. Failure to properly remove the circuit boards or interposers can result in permanent damage to the circuit assemblies.***

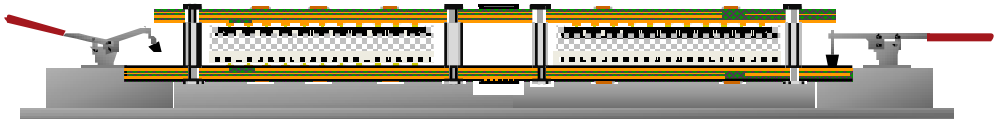
Unless specifically mentioned, IT3 and IT5 share the same capability and requirement.

13.1 Daughter Card Removal

To remove a daughter card, first remove the nuts from the reinforcing spacers, then lift the daughter card straight off the interposers, as shown:

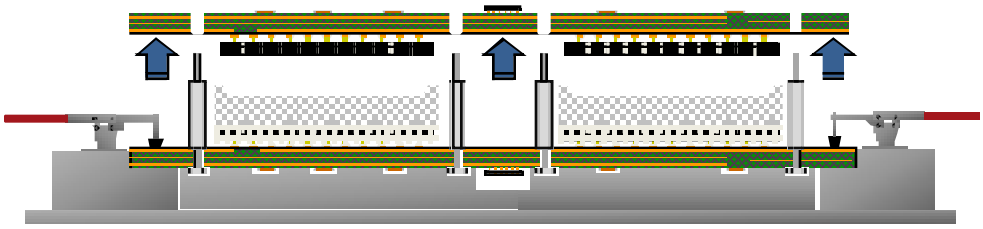


Precaution: It is **very important to prevent excessive flexing** of the circuit assemblies during disassembly operations. To minimize flexing of the mother board, a simple **tooling plate** is suggested:



The tooling plate has **clamps to stabilize the mother board** while the daughter card (and possibly the interposer) is removed.

The daughter card should be **lifted straight up** off the interposers.

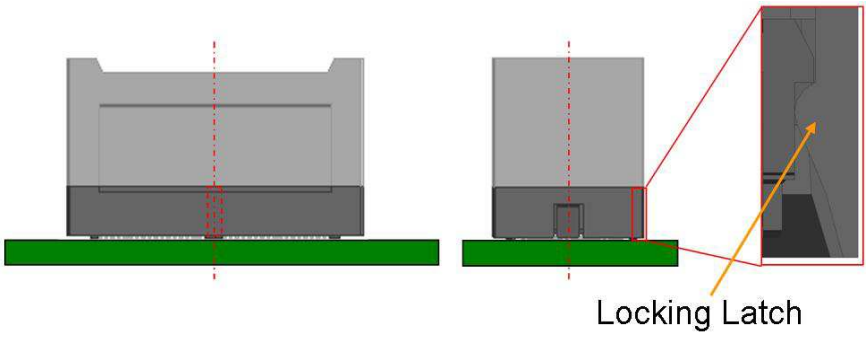


To **minimize unnecessary flexing** of the daughter card, the removal **forces should be applied as close to the interposer as possible** without contacting any components. On densely populated assemblies, the edges may be the only open area that can be grasped.

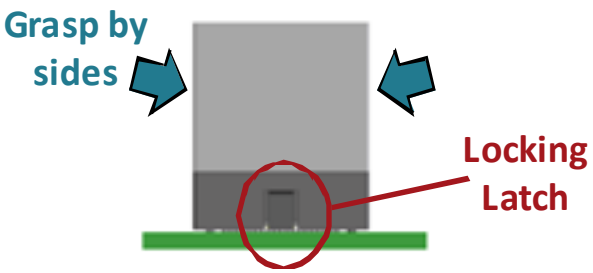
13.2 Interposer Removal

The IT3 interposer is secured onto the mounting receptacle with snap fit tabs, shown below:

The IT3 interposer is secured onto the mounting receptacle also with other snap fit tabs, shown below. The IT5 interposer uses this mechanism solely.



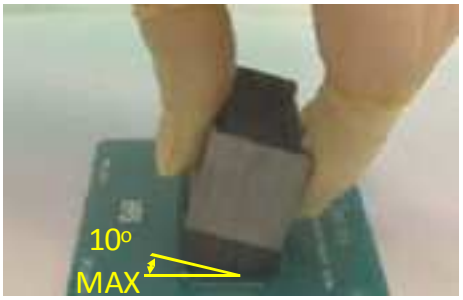
For removal, interposer should be grasped by these sides as shown. They do not have locking latches.



Interposer Removal by Hand

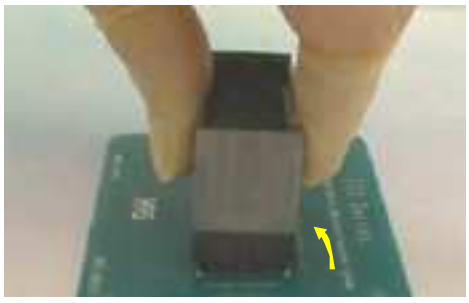
1) Hold the Interposer Assembly on the walls without IT3 locking latches

2) Gently rotate one side of the Interposer Assembly laterally 10° maximum

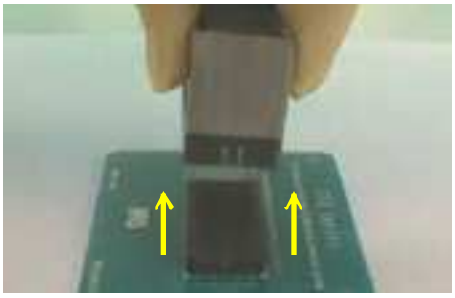


Caution: do not rotate more than 10 degrees

3) While gently rotating, pull up on other side of the Interposer Assembly



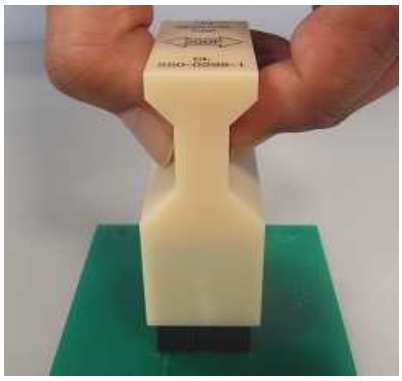
4) The Interposer Assembly is removed from the board assembly, and the Mounting Receptacle is ready to accept another Interposer Assembly



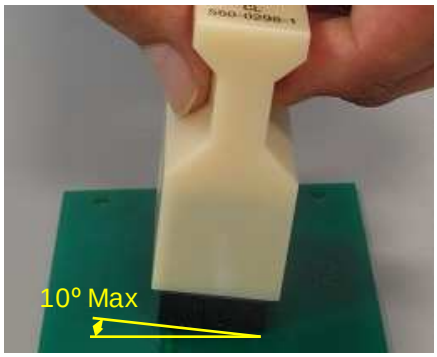
An interposer removal tool is also available. **This tool is not an interposer installation cap, so please do not use it to install an interposer.** Doing so may damage an interposer.

Interposer Removal with Tool

1) Cover the Interposer Assembly with the interposer removal tool

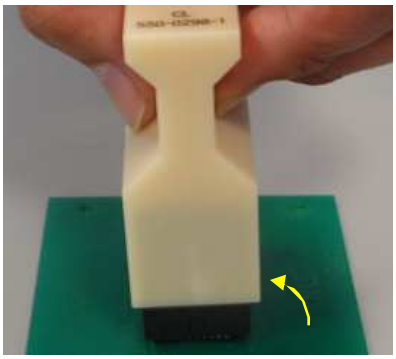


2) Gently rotate one side of the Interposer Assembly laterally 10° maximum using the tool

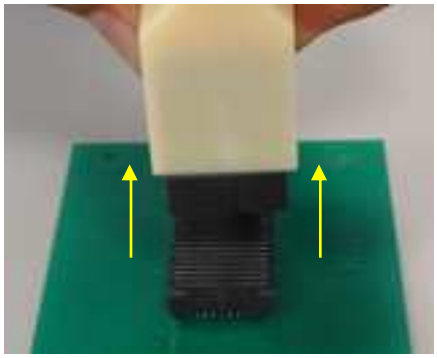


Caution: do not rotate more than 10 degrees

3) While gently rotating, pull up on other side of the tool



4) The Interposer Assembly is removed, as it is inside the tool



Precaution: Visually inspect the interposer before reinstalling it. Discard if it shows any signs of damage or wear. Do not subject interposer assembly to more than five removal-reinstallation cycles, even if it appears unaffected.

HIROSE ELECTRIC (U.S.A.), INC.
High Speed Interconnects
3255 Scott Blvd., Building 7, Ste. 101, Santa Clara, CA 95054, U.S.A.
www.hiroseusa.com