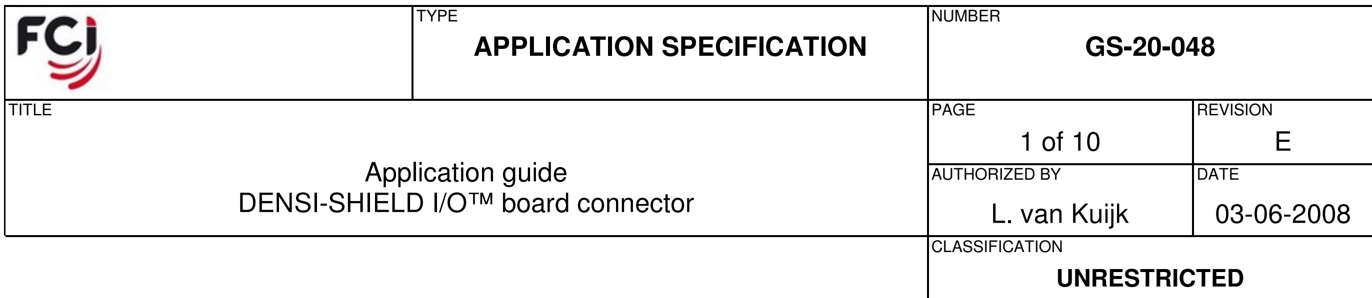


TYPE: APPLICATION SPECIFICATION	NUMBER: GS-20-048	
TITLE: Application guide DENSI-SHIELD I/O™ board connector	PAGE: 1 of 10	REVISION: E
	AUTHORIZED BY: L. van Kuijk	DATE: 03-06-2008
	CLASSIFICATION: UNRESTRICTED	

1 OBJECTIVE....2

2 SCOPE....2

3 GENERAL....2

4 DRAWINGS AND APPLICABLE DOCUMENTS....2

5 APPLICATION REQUIREMENTS....2
5.1 PCB layout....2
5.2 Keep-out zone....2
5.3 Packaging....2
5.4 Multiple connector applications....3

6 APPLICATION TOOLING....3

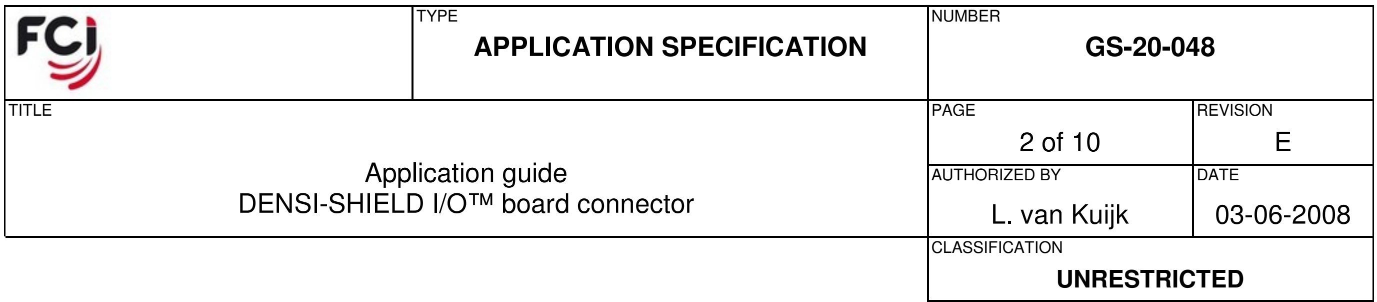
7 APPLICATION PROCEDURE....3
7.1 Connector placement....3
7.2 Solder paste application....3
7.3 Example of solder reflow....4
7.4 Cleaning....5
7.5 Inspection....5

8 REWORK....6
8.1 Rework application tooling....6
8.2 Rework procedure....6
8.3 Rework procedure vapour phase....7

9 REVISION RECORD....8

10 APPENDIX 1: DRAWINGS OF THE PARTS OF DESOLDERING TOOL....9

Copyright FCI



1 OBJECTIVE

This specification provides information and requirements for customer application of the DENSI-SHIELD I/O™ board connector. It is intended to provide general guidance for process development. It should be recognized that no single process will work under all customer applications and that customers should develop processes to meet individual needs. However, if the process varies greatly from the recommended one, FCI cannot guarantee acceptable results.

2 SCOPE

This specification provides information and requirements regarding application of the DENSI-SHIELD I/O™ board connector (FCI product number: 10044471) to printed circuit boards (PCB).

3 GENERAL

This document is a general application guide. If there is a conflict between the product drawings and this specification, the drawings take precedence.

4 DRAWINGS AND APPLICABLE DOCUMENTS

FCI product drawings and specifications are available by accessing the FCI website or contacting the FCI Technical Service. In the event of a conflict between this specification and the product drawing, the drawing takes precedence. Customers should refer to the latest revision level of FCI product drawings for appropriate product details.

5 APPLICATION REQUIREMENTS

To insure proper application, several factors must be considered before the solder reflow step including PCB design/quality, proper application of solder paste and placement of the connectors on the PCB.

The flatness of the PCB and the solder lands are essential to placement quality.

5.1 PCB LAYOUT

For recommended PCB- and stencil layout refer to the customer drawing of the part number being applied.

5.2 KEEP-OUT ZONE

For recommended keep-out zone around the part perimeter refer to the customer drawing of the part number being applied.

5.3 PACKAGING

The available packaging method is tape-and-reel.

Copyright FCI

5.4 MULTIPLE CONNECTOR APPLICATIONS

The connector has no holding features, so the connector should be placed on the topside of the PCB, or a holding fixture should be used to insure the connector stays in its position during reflow.

6 APPLICATION TOOLING

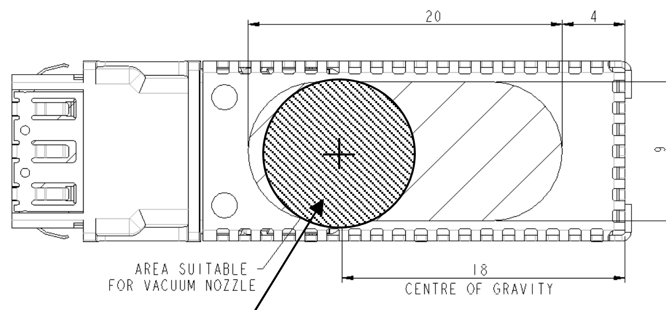
No specific application tooling is needed for this connector.

7 APPLICATION PROCEDURE

7.1 CONNECTOR PLACEMENT

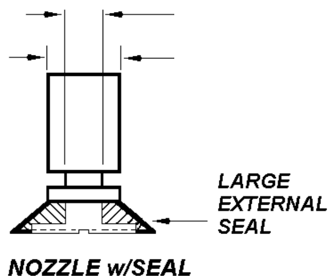
Connectors are packaged and shipped in tape-and-reel. Placement is possible by hand or with a mechanical gripper or with a vacuum nozzle.

To allow for the proper use of a vacuum nozzle, the board connector has a flat surface of sufficient area parallel to the board-mounting plane.



Nozzle pick-up position.

Recommended nozzle type for 8p connector



7.2 SOLDER PASTE APPLICATION

The volume of solder paste as well as its position accuracy is critical to the placement quality to insure a strong, consistent solder joint.

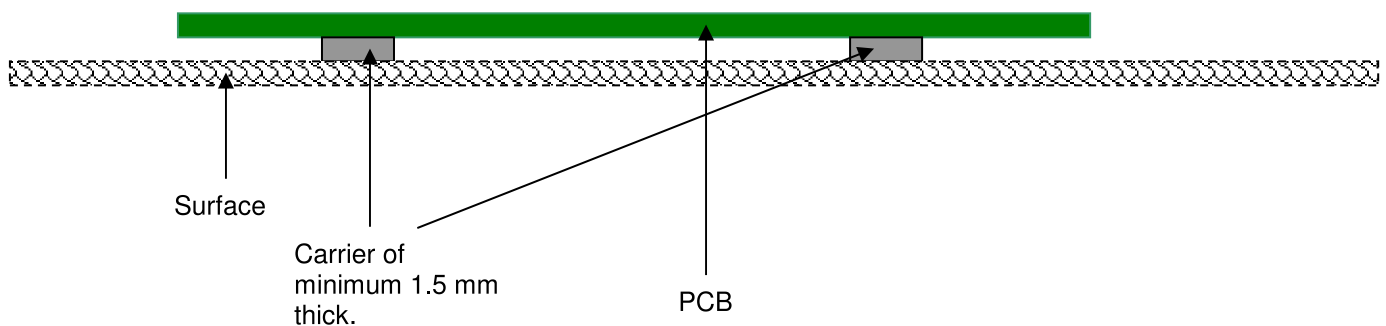
Copyright FCI

The solder content in the paste should be sufficient. The screen-printing process (angle, speed, pressure, etc.) must be optimized so that enough solder is available.
For Pin-in-paste recommendations, see the FCI website; www.FCI.com under; Documentation, the file "Pin-in-Paste Application Guide"

A no-clean solder paste is required.

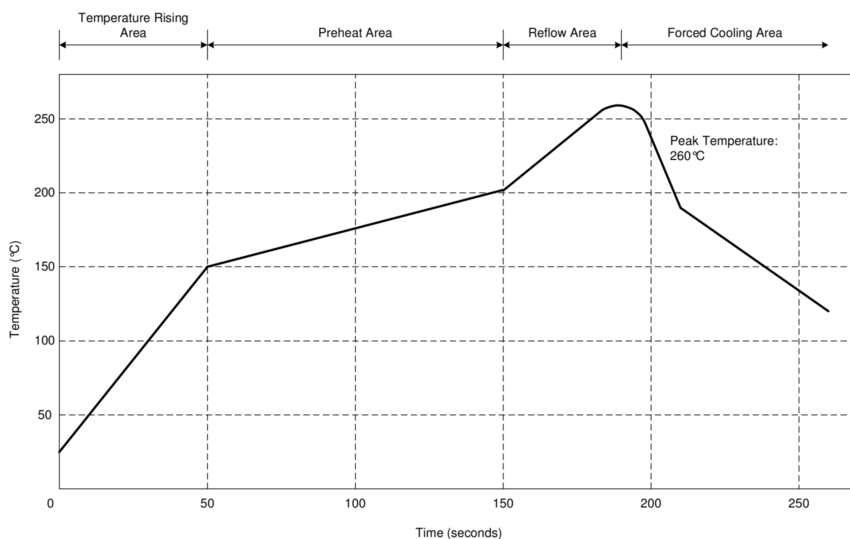
A high quality, laser cut stencil is recommended. For stencil design and thickness refer to IPC-7525 Stencil Design Guidelines and to the customer drawing of the part number being applied.

During the screen-printing and reflow process PCB's thinner than 2.1 mm should be supported by a carrier of minimum 1.5 mm thick because the solder tails and pegs of the board connector could protrude from the underside of the PCB. The carrier should not touch these pegs and solder tails.



7.3 EXAMPLE OF SOLDER REFLOW

- Solder paste
  - Name : Hi performance lead free solder paste S3X58-M405
  - Supplier : Koki Company Limited
  - Type : Sn96.5, Ag3.0, Cu0.5
- Reflow profile : See profile below
  - Preheating condition (temperature rising & preheat area)
    - Peak temperature : max. 200 °C (+0/-5 °C)
    - Soak time : 120 – 200 s
    - Average temperature gradient : $\Delta T/\Delta t$ = 2.5 – 4.9 °C/s
  - Temperature gradient in reflow : $\Delta T/\Delta t$ = max. 3 - 5 °C/s
  - Time above 200 °C : 60 – 90 s
  - Time above 217 °C : 30 – 60 s
  - Time above 230 °C : 15 – 50 s
  - Time above 250 °C : 10 – 20 s
  - Peak temperature : max. 260 °C (+0/-5 °C)
  - Temperature gradient in cooling : $\Delta T/\Delta t$ = max. -6 °C/s
  - Total reflow profile duration : 250 – 400 s from 40 °C in preheating to 100 °C in cooling
  - Total cool down time : min. 5 minutes

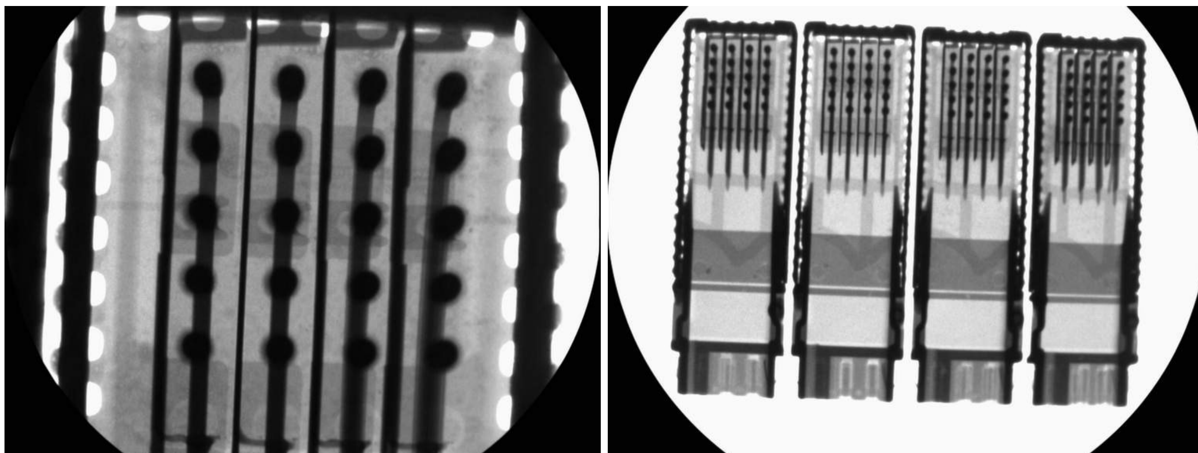


7.4 CLEANING

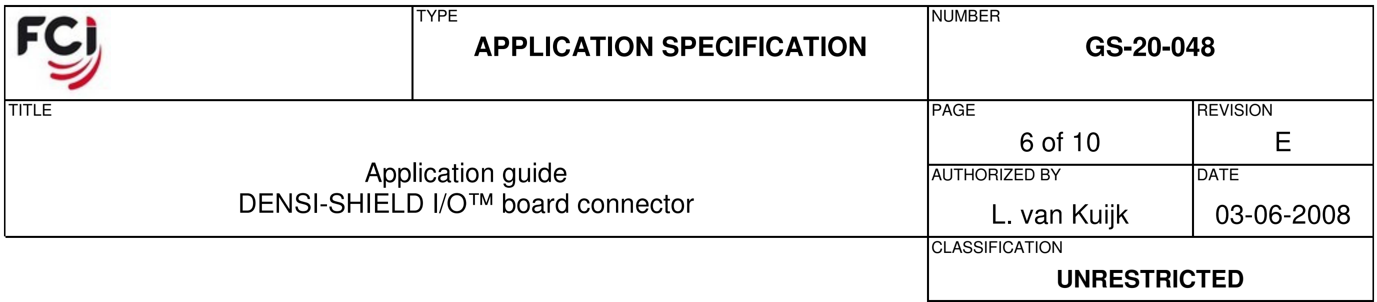
A connector is an electro-mechanical component of which the functioning is highly affected by contamination. Therefore, cleaning with solvents is not allowed. The chance of spreading the contamination onto the contact area should be minimized. The use of a no clean flux is required.

7.5 INSPECTION

Visual inspection of the connector for damage and cleanliness is recommended.
X-ray technology can be used to check for solder balls and solder voids.



Electrical testing for opens and shorts is recommended.



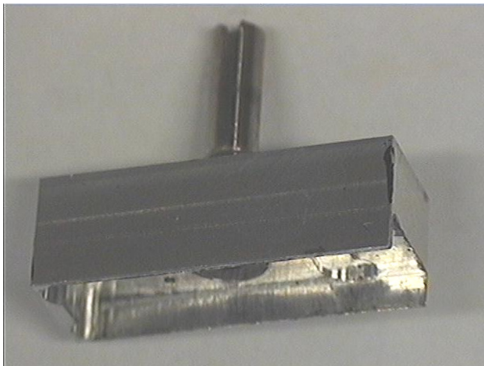
8 REWORK

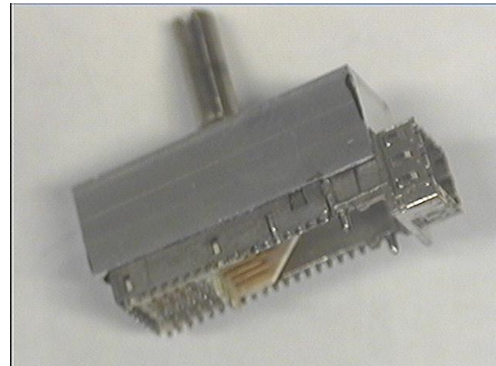
8.1 REWORK APPLICATION TOOLING

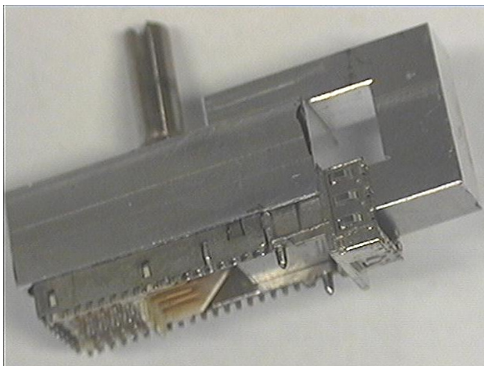
Product number	Tool description	Remark
	Desoldering/repair tooling	See appendix 1

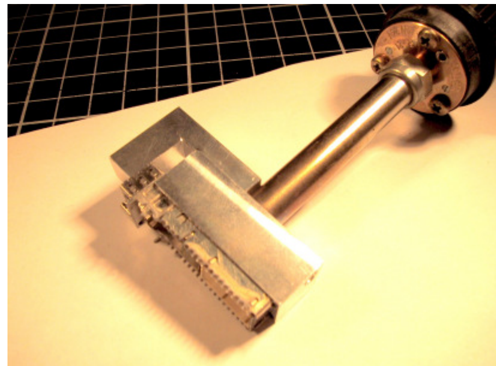
8.2 REWORK PROCEDURE

First remove the cover by heating it up to a temperature higher than the melting point of the used solder.
A solder iron with a special head can be used for this operation.









Note that it will take some time before the cover is on the right temperature.
To prevent damage to the PCB care must be taken to pull off the connector in a straight line with a constant force not larger than 10N.

The IMLA's can be removed by blowing hot air (with a heatgun) under the PCB along the solder pads.

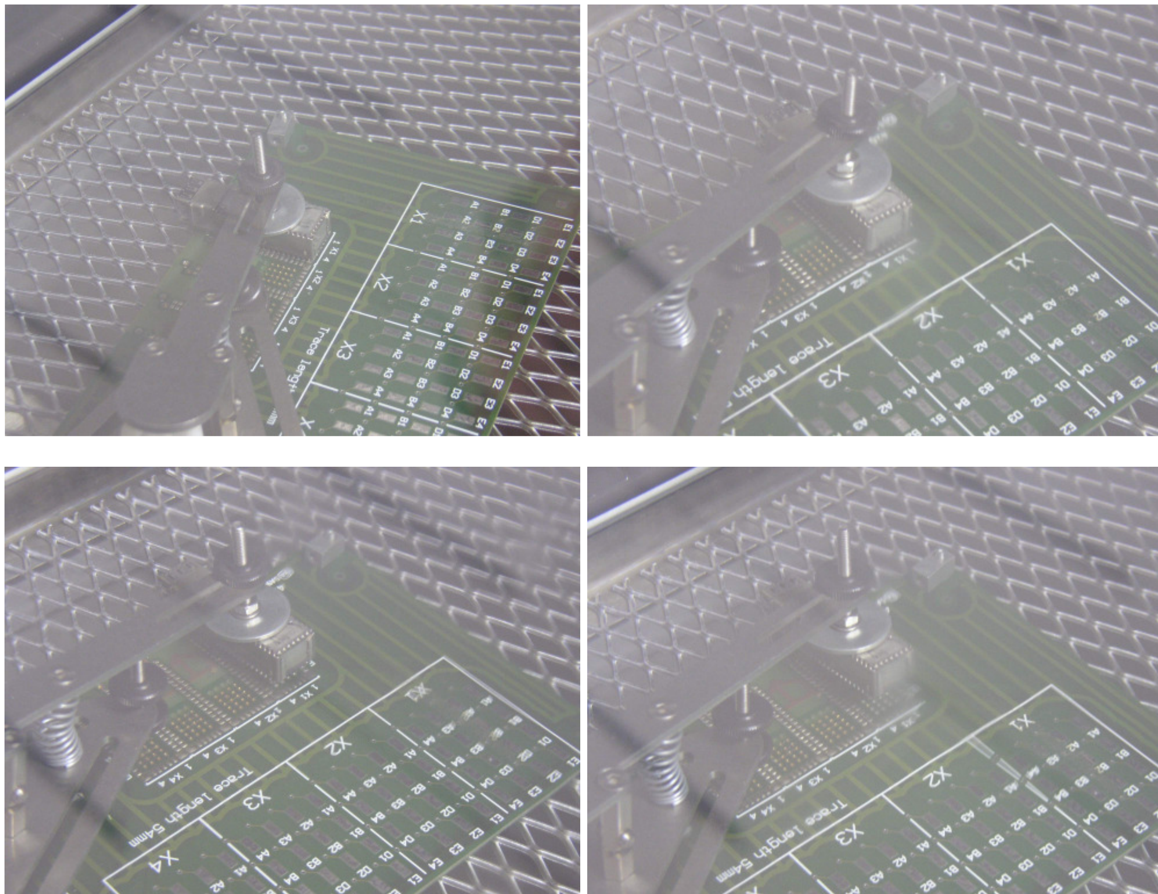
Copyright FCI

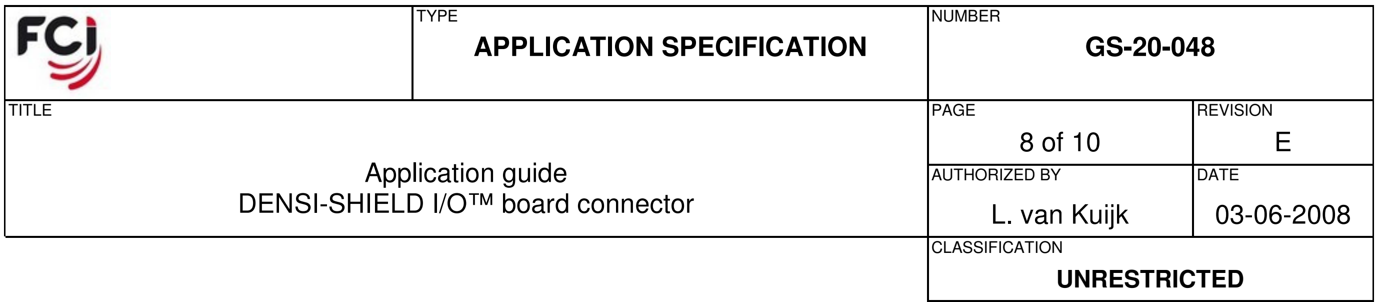
It is possible to remove the complete connector in one step when the cover and the solder pads are heated up at the same time.

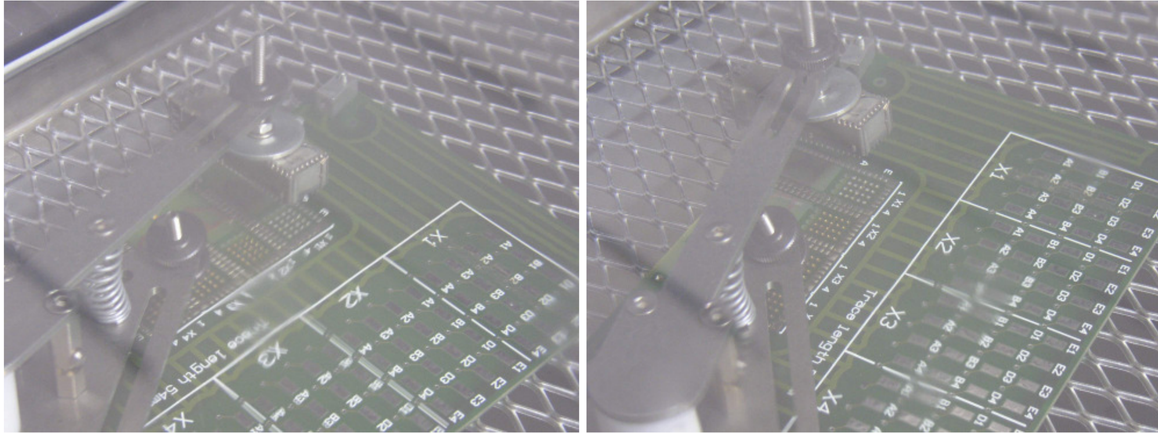
After the removal of the cover and IMLA's, clean the PCB by removing excess solder and apply new solder paste.
Place a new connector and reflow the PCB.

8.3 REWORK PROCEDURE VAPOUR PHASE

The connector can be removed easily with the use of vapour phase techniques and standard tooling.







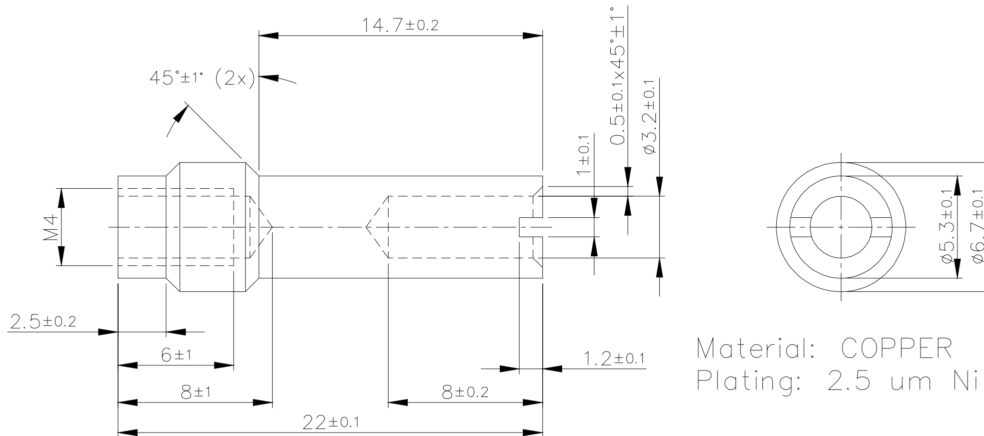
9 REVISION RECORD

REV	PAGE	DESCRIPTION	ECR#	DATE (dd-mm-yy)
A	All	New release		27-05-2005
B	All	Updated Logo & Footer	H06-0100	10/07/2006
C	9-10	Added drawings repair tool	H07-0125	12-10-2007
D	4	Reference to Pip application guide	H08-0057	22-04-2008
E	4	Reference to Pip application guide on website & guardian trans.	H08-0066	25-02-2009

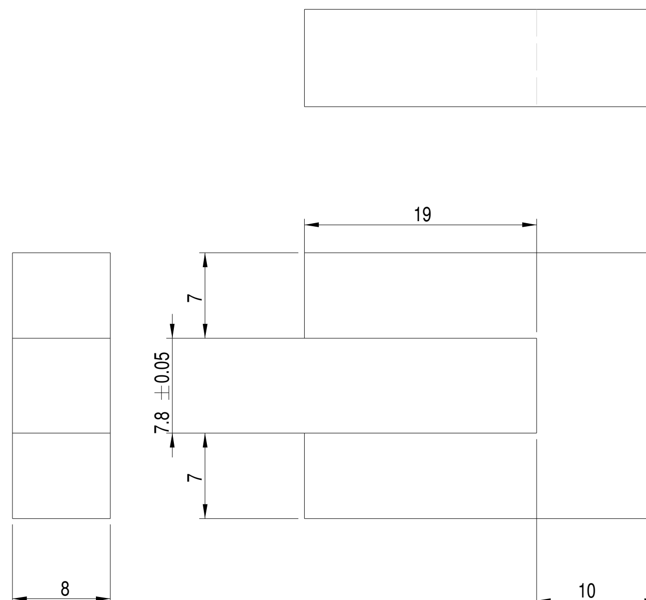
Copyright FCI

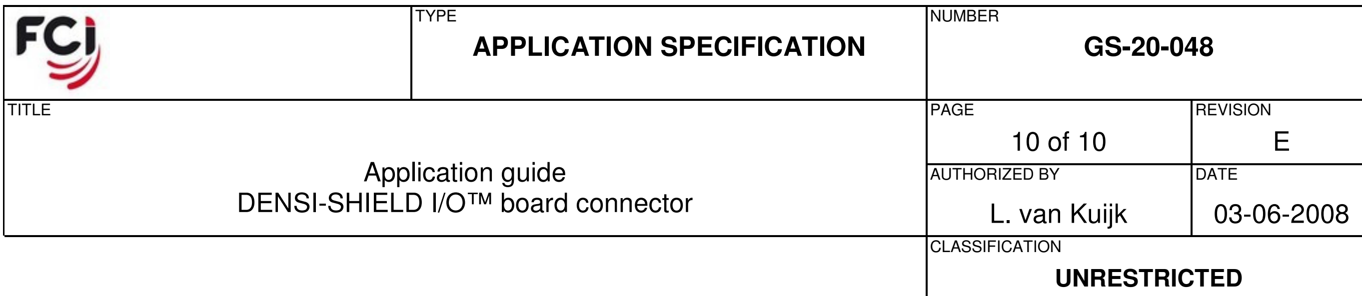
10 APPENDIX 1: DRAWINGS OF THE PARTS OF DESOLDERING TOOL

Part 1: Shaft between Weller WS50 solder station and heating block. When using other solder station this part should be adapted (Suggestion: Adapt solder tip).

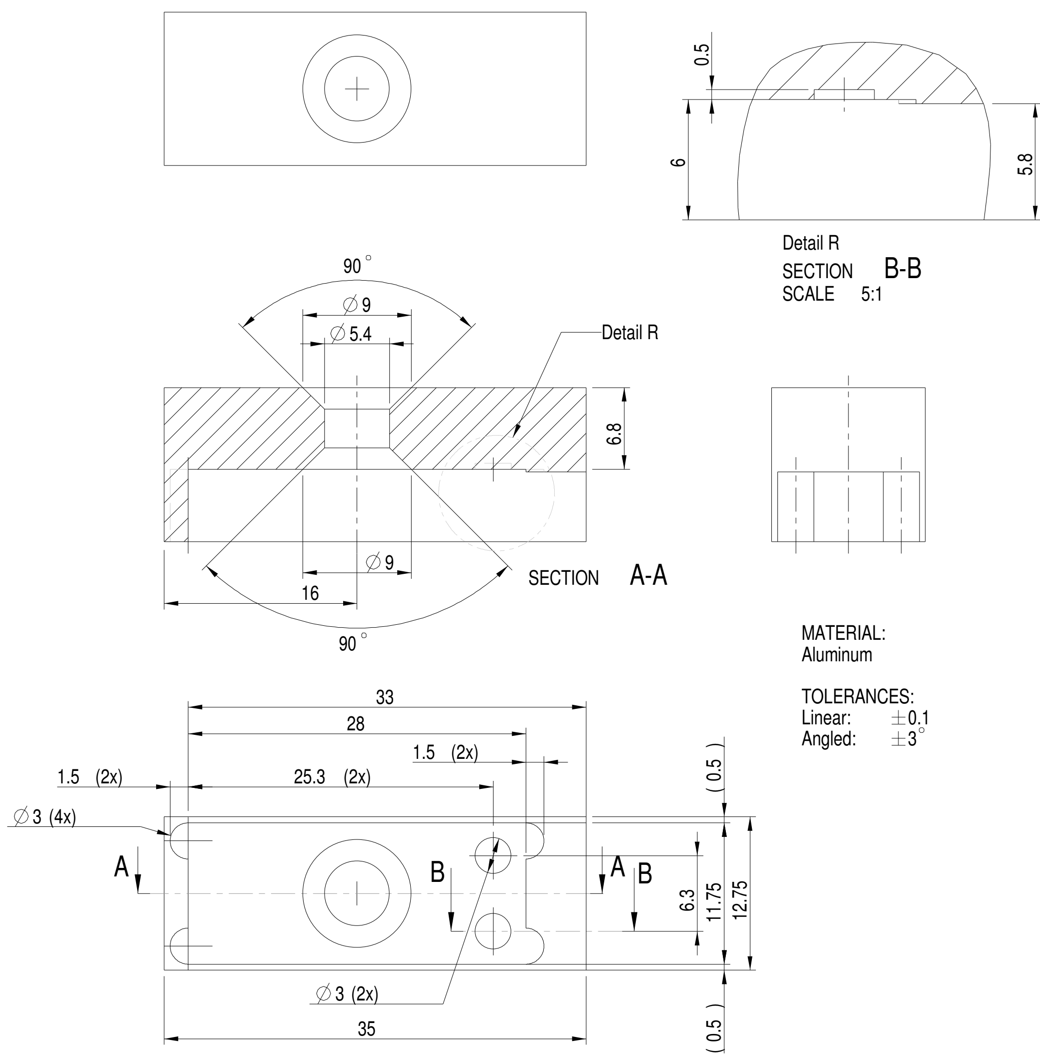


Part 2: BC holder





Part 3: Heating block



Copyright FCI