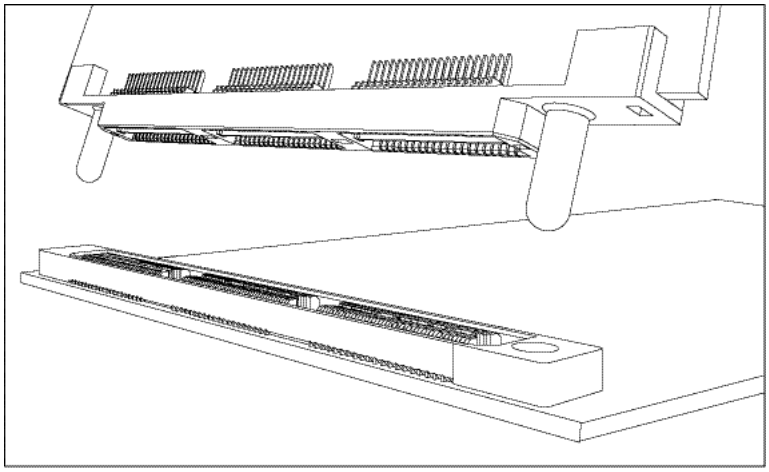
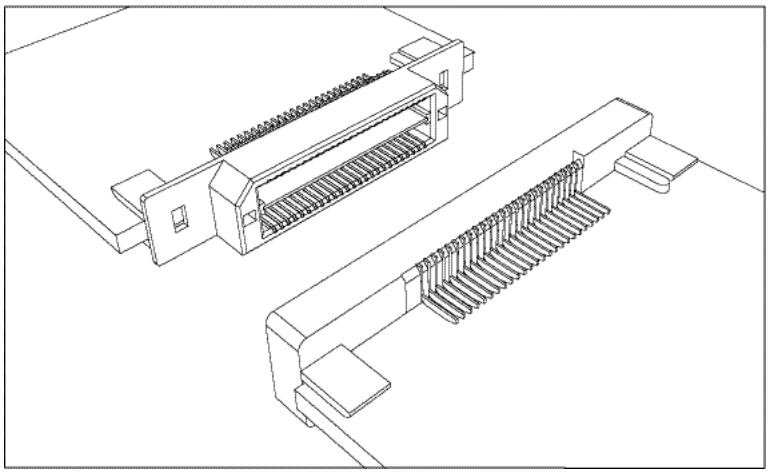
EDGE MOUNT Q-STRIP™/Q-PAIRS™ INTERCONNECTS

APPLICATION OVERVIEW

Samtec edge-mount Q-Strip™ and Q-Pairs™ interconnects have been designed for perpendicular (90°) and coplanar (180°) Board-to-Board applications for .062" thick PC boards. The integral ground plane between the signal rows helps to maximize signal integrity by minimizing impedance, cross-talk, attenuation and VSWR. Common applications include:

- Servers
- Routers
- Set top boxes
- Network systems
- Switchers
- Hubs
- Bridges
- Industrial computers
- Test systems
- System upgrades





Perpendicular and coplanar applications

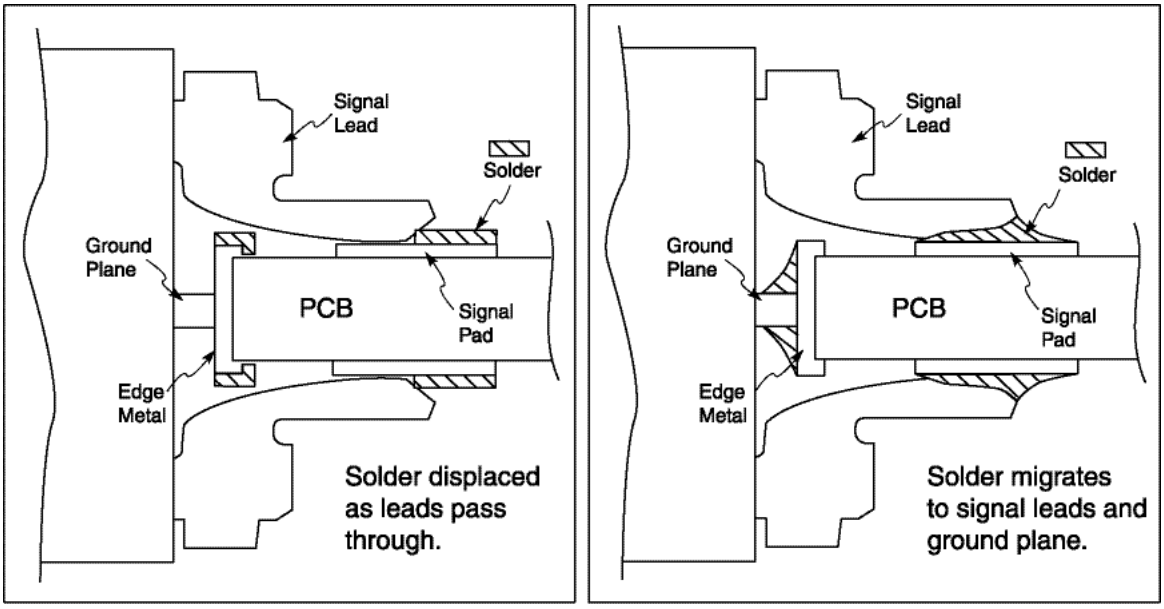
These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING OVERVIEW

Samtec's hi-speed Q-Strip™ and Q-Pairs™ edge mount interconnects can be used to connect PC boards (PCBs) populated on one or both sides of the board.The following guidelines were developed for Samtec's Q-Strip™ and Q-Pairs™ interconnect systems on .5mm, .635mm, and .8mm centerlines. Specifically, these guidelines are relevant for Samtec's QTH/QSH, QTS/QSS, and QTE/QSE series connectors with edge mount leads on .062" thick PC boards.

Other than the micro pitch signal leads, the integrated ground planes within these connectors can pose a challenge from a processing standpoint. The following guidelines provide step-by-step procedures to help ensure that acceptable solder joints are achieved for the integrated ground planes as well as the edge-mount signal leads.

While we have made every attempt to cover most situations, these guidelines and suggestions should not be considered design requirements for all applications. Samtec highly recommends testing the Q-Strip™ and Q-Pairs™ edge-mount interconnects on your boards and in your process to guarantee optimum results.



Before and after re-flow.

Three separate sets of processing guidelines are provided in the following pages: Single sided PC boards (SMT components populated on one side of PCB only), Dual sided PC boards using .8mm pitch connectors (SMT components populated on both sides of PCB), and Dual sided PC boards using .5mm and .635mm pitch connectors. Please refer to the correct set of processing guidelines for your particular design.

These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING GUIDELINES – SINGLE SIDED PCB

(SMT components populated on one side of PCB only)

General Processing Equipment and Conditions:

- .006"(0,15mm) stencil
- Chemical or laser etched stencils
- Oven speed/temperature settings within typical SMT process boundaries
- Connectors hand-placed on the PCB
- PCB Edge Metal is to be HASL plated (strongly recommended)

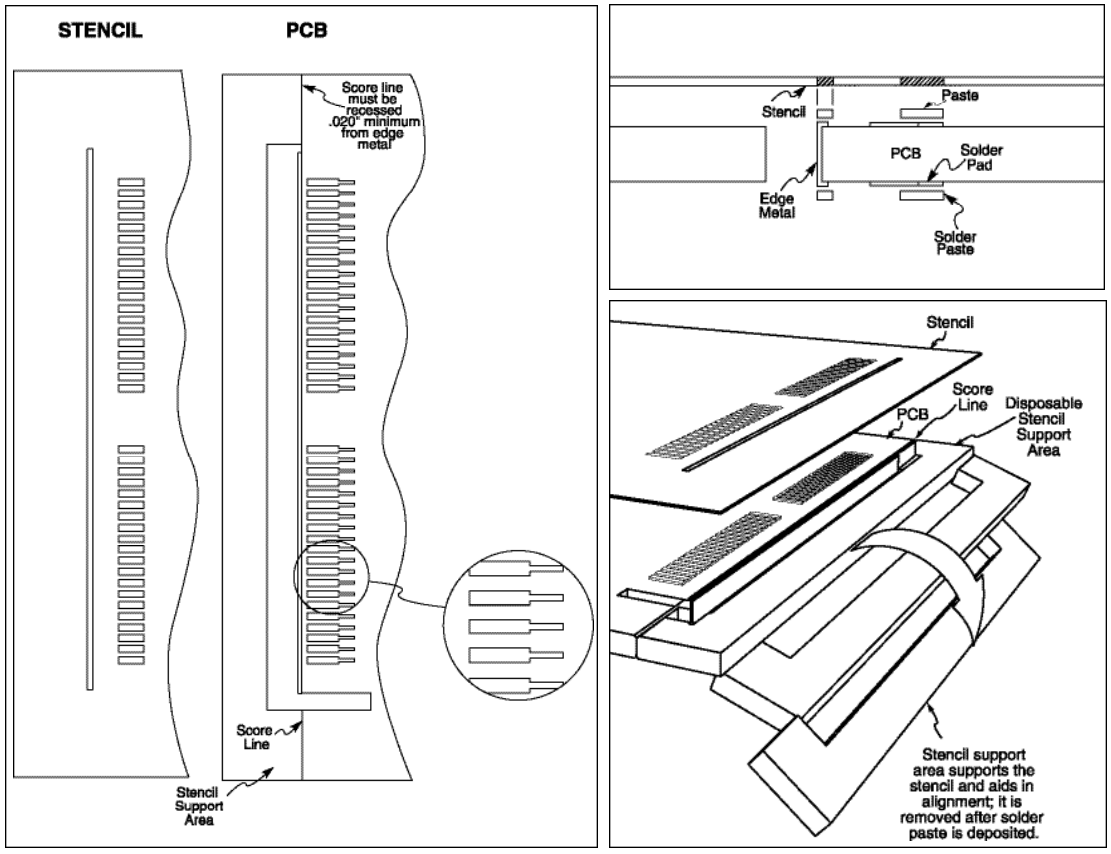
Recommendations:

- **Alignment features** for proper lead/pad registration are strongly recommended.

- Samtec recommends a **stencil "support" area** on the PCB. This area supports the stencil while solder paste is screened on the PCB, and it aids in the alignment of the stencil and PCB.

- Score lines* separate the stencil support area from the populated section of the PCB. After solder paste is deposited on the PCB, the stencil support area is removed and discarded.The edge mount interconnect is then attached and soldered to the PCB.

  ** Score Line: The score line is a "V" shaped cut on opposing sides of the PCB used to divide up multiple boards that can be separated easily by manually bending and breaking it.*



***** NOTE:The above diagram is for reference only. Please refer to the official Samtec recommended stencil and PCB layout drawings for the specific connector series that you will be using (QTH, QSH, QTS, QSS, QTE, or QSE series). PDFs may be downloaded from page 12 of this processing guide.*

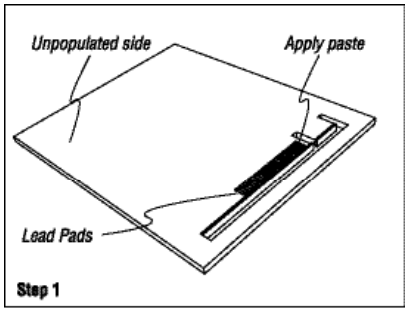
These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING GUIDELINES – SINGLE SIDED PCB (cont.)

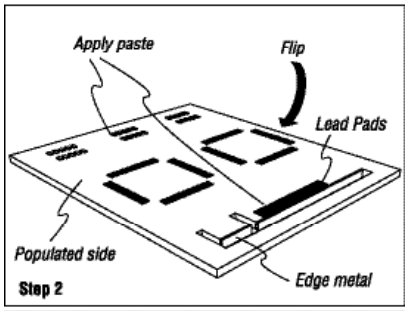
(SMT components populated on one side of PCB only)

Procedures:

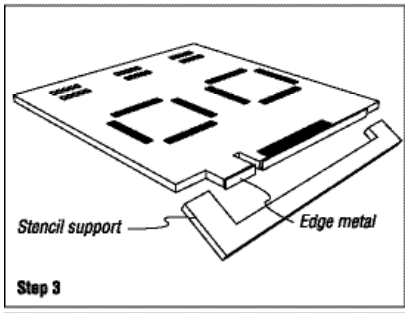
1. Apply solder paste to the unpopulated side of the PCB. Solder paste is deposited on the lead pads and on the **top side of the ground plane pad**. Do not deposit solder directly on the edge of the PCB/**edge metal**.



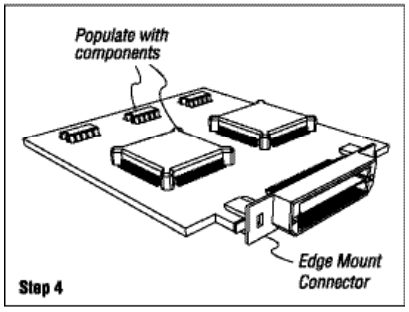
2. Flip and apply solder paste to the populated side of the PCB. Solder paste is deposited on the lead pads and on the top side of the ground plane pad. Do not deposit solder directly on the edge of the PCB (edge metal).



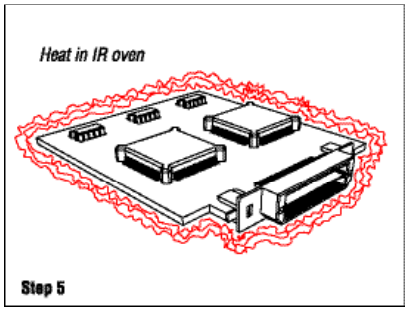
3. Remove the stencil support area of PCB and discard.



4. Place edge mount connector on PCB at the same time other components are placed on PCB.



5. Process in IR oven using standard SMT re-flow parameters.



These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING GUIDELINES – DUAL SIDED PCB .5mm & .635mm

(SMT components populated on both sides of PCB)

General Processing Equipment and Conditions:

- .006"(0,15mm) stencil
- Chemical or laser etched stencils
- Oven speed/temperature settings within typical SMT process boundaries
- Connectors hand-placed on the PCB
- PCB Edge Metal is to be HASL plated (strongly recommended)

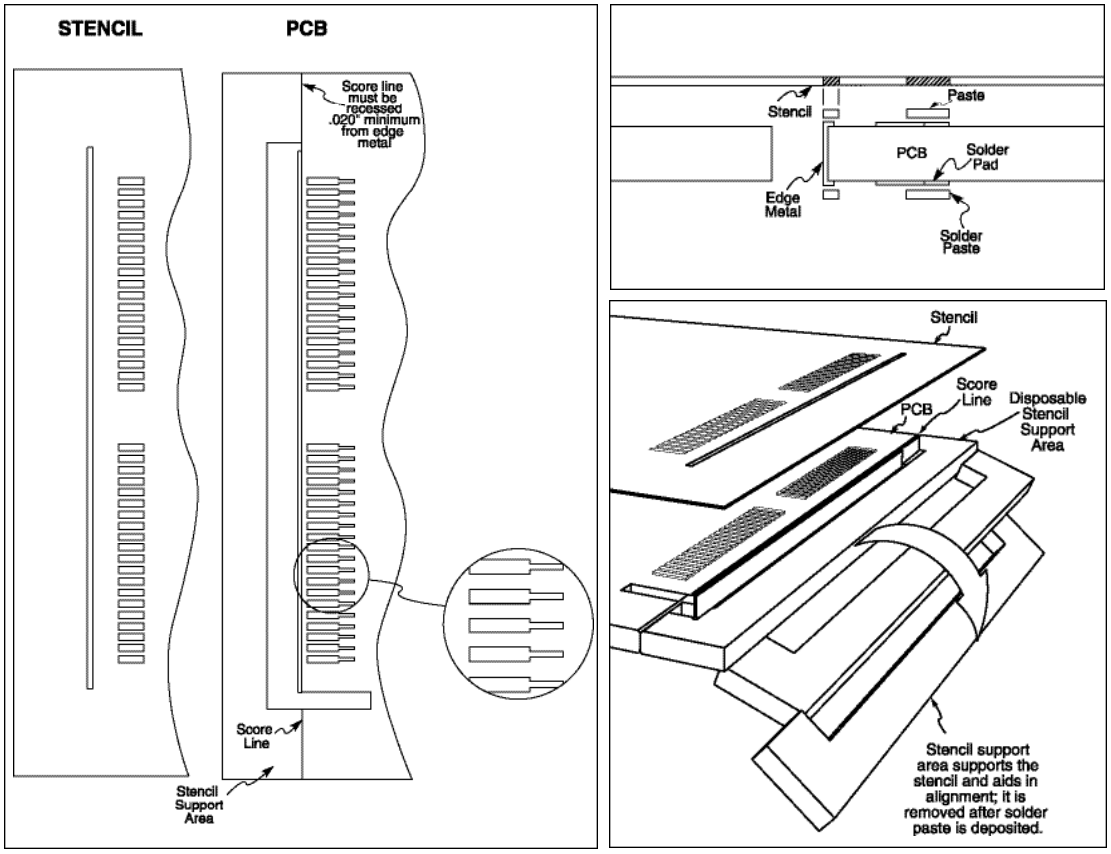
Recommendations:

- **Alignment features** for proper lead/pad registration are strongly recommended.

- Samtec recommends a **stencil "support" area** on the PCB. This area supports the stencil while solder paste is screened on the PCB, and it aids in the alignment of the stencil and PCB.

- Score lines separate the stencil support area from the populated section of the PCB. After solder paste is deposited on the PCB, the stencil support area is removed and discarded.The edge mount interconnect is then attached and soldered to the PCB.

** Score Line: The score line is a "V" shaped cut on opposing sides of the PCB used to divide up multiple boards that can be separated easily by manually bending and breaking it.*



***** NOTE:The above diagram is for reference only. Please refer to the official Samtec recommended stencil and PCB layout drawings for the specific connector series that you will be using (QTH, QSH, QTS, QSS, QTE, or QSE series). PDFs may be downloaded from page 12 of this processing guide.*

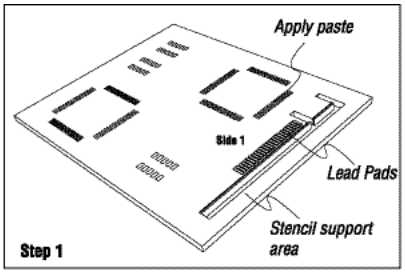
These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING GUIDELINES – DUAL SIDED PCB .5mm & .635mm (cont.)

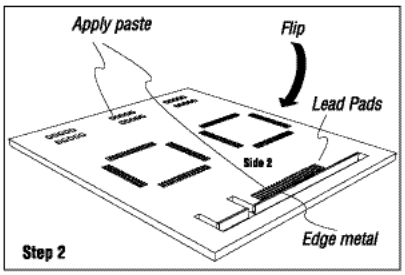
(SMT components populated on both sides of PCB)

Procedures:

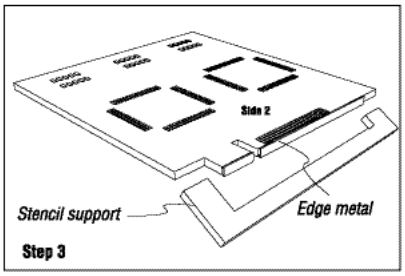
1. Apply solder paste to the edge mount lead pads and to the **top of the ground plane pad,** on one side of PCB only. Do not deposit solder on the edge of the PCB/**edge metal**.



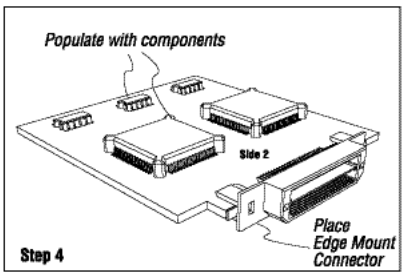
2. Flip PCB and apply solder paste to the edge mount lead pads, to the top of the ground plane pad, and to the rest of side two of the PCB. Do not deposit solder on the edge of the edge metal.



3. Remove the stencil support area of PCB and discard.



4. Place edge mount connector on PCB at the same time other components are placed on side two.



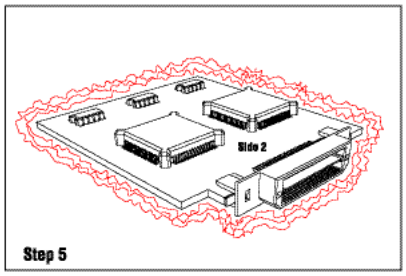
These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING GUIDELINES – DUAL SIDED PCB .5mm & .635mm (cont.)

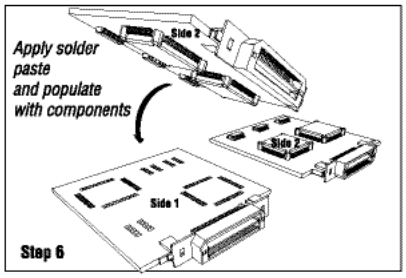
(SMT components populated on both sides of PCB)

Procedures:

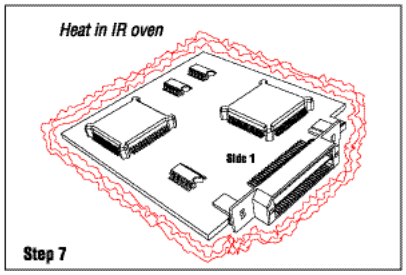
5. Process using standard SMT reflow parameters.



6. Flip PCB and apply solder paste to side one of PCB and populate with remaining components.



7. Process using standard SMT reflow parameters.



These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING GUIDELINES – DUAL SIDED PCB .8mm

(SMT components populated on both sides of PCB)

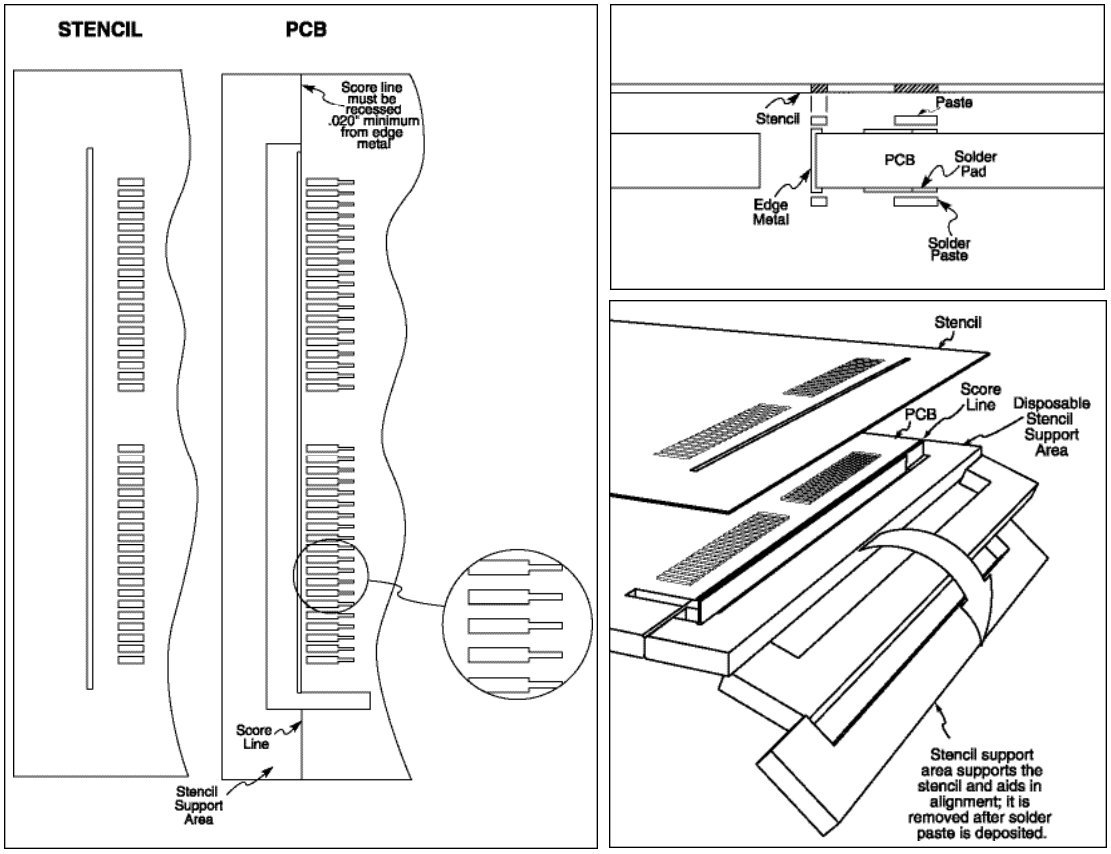
General Processing Equipment and Conditions:

- .006"(0,15mm) stencil
- Chemical or laser etched stencils
- Oven speed/temperature settings within typical SMT process boundaries
- Connectors hand-placed on the PCB
- PCB Edge Metal is to be HASL plated (strongly recommended)

Recommendations:

- **Alignment features** for proper lead/pad registration are strongly recommended.
- Samtec recommends a **stencil "support" area** on the PCB. This area supports the stencil while solder paste is screened on the PCB, and it aids in the alignment of the stencil and PCB.
- Score lines separate the stencil support area from the populated section of the PCB. After solder paste is deposited on the PCB, the stencil support area is removed and discarded.The edge mount interconnect is then attached and soldered to the PCB.

** Score Line: The score line is a "V" shaped cut on opposing sides of the PCB board used to divide up multiple boards that can be separated easily by manually bending and breaking it.*



***** NOTE:The above diagram is for reference only. Please refer to the official Samtec recommended stencil and PCB layout drawings for the specific connector series that you will be using (QTH, QSH, QTS, QSS, QTE, or QSE series). PDFs may be downloaded from page 12 of this processing guide.*

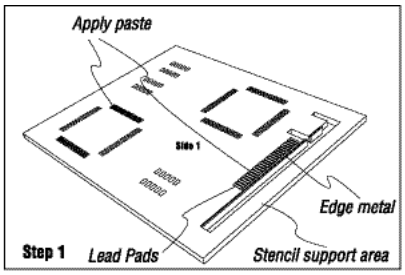
These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING GUIDELINES – DUAL SIDED PCB .8mm (cont.)

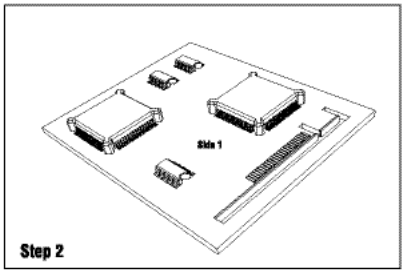
(SMT components populated on both sides of PCB)

Procedures:

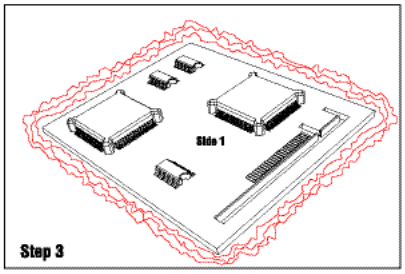
1. Apply solder paste to the edge mount lead pads, to the top of the ground plane pad and to the rest of side one of the PCB. Do not deposit solder on the edge of the edge metal.



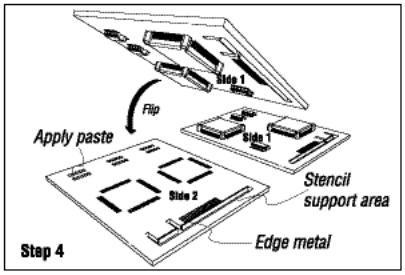
2. Place SMT components on side one. Do not place edge mount connector on PCB at this time.



3. Process using standard SMT reflow parameters.



4. Flip PCB and apply solder paste to the edge mount lead pads, to the top of the ground plane pad and to the rest of side two of the PCB. Do not deposit solder on the edge of the edge metal.



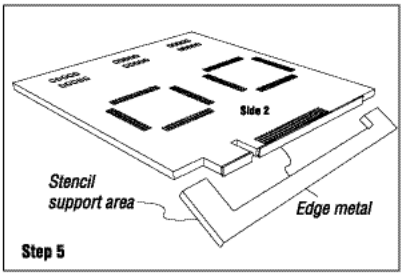
These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

PROCESSING GUIDELINES – DUAL SIDED PCB .8mm (cont.)

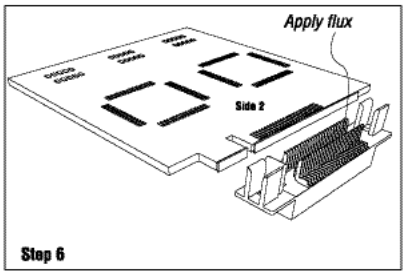
(SMT components populated on both sides of PCB)

Procedures:

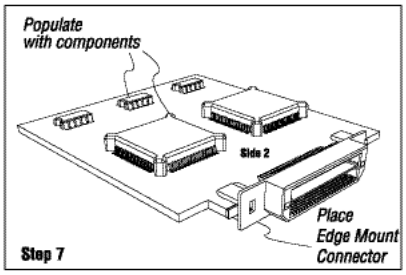
5. Remove the stencil support area of PCB and discard.



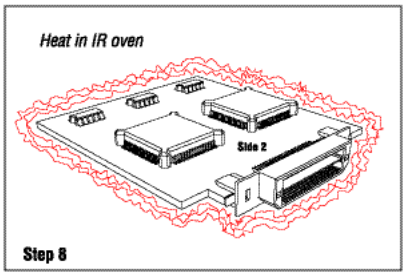
6. Apply flux to leads to be soldered to side one of the PCB.



7. Place edge mount connector on PCB at the same time other components are placed on side two.



8. Process using standard SMT reflow parameters.



These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

FREQUENTLY ASKED QUESTIONS

Q. How do I minimize solder bridging on the edge-mount tails?
A. Due to the direction of application of the edge-mount connector, the solder paste can wipe sideways and cause bridging to the neighboring contacts. You can minimize it by designing in the special pad with the skinny "candle-wick" on the end of the pad. It helps hold the solder and guide the paste during re-flow towards the solder tail.

Q. Why does the solder stencil seem to be offset from the pad?
A. See above answer.

Q. How does the ground plane get soldered?
A. The .015"X .006" thick solder paste applied on the top and bottom edge of the board flows towards the hottest spot which will be the ground plane touching the edge of the board.

Q. Will these interconnects work on a .031" or .093" thick PC Board?
A. No, these interconnects are designed for use on .062" thick PC boards.

Q. Will a PCB coated with OSP (organic solderability preservative) work with Samtec's high-speed edge-mount connectors?
A. At this time Samtec has not successfully used OSP due to its resistance to solder flow, which is required to form an acceptable solder joint. Samtec does not recommend OSP due to additional processing variability, and recommends consulting with your manufacturing staff prior to using OSP coating in this application.

Q. How can I process these edge-mount connectors to my boards, which are not plated using HASL (Hot Air Solder Leveling)?
A. Some PCB plating methods other than HASL can accept Samtec's edge-mount connectors. However, it may be a requirement to apply the solder paste directly to the ground plane edge-metal or to the connector ground plane itself. If your PCB is not plated using HASL, please contact Samtec's Interconnect Processing Group to discuss your specific application.

Q. Will gold plated tails and HASL plating with Sn/Pb solder cause solder joint imbrittlement?
A. Due to the small geometry of the contacts and terminals, duplex plating is not an available option. In the history of Samtec, we have not seen evidence of or actual cases of solder joint imbrittlement due to intermetallic properties.

Q. How do the footprint layouts of Samtec's Q-Pairs™ (QXX-EM-DP, differential pair) connectors differ from the regular Q-Strip™ footprint layouts (QXX-EM)?
A. The -DP style has every third position pad removed. Otherwise all other dimensions apply.

Q. Why are score lines stepped back from the edge of the edge-metal?
A. The score line is necessary when a stencil is used and a stencil support is necessary. It is stepped back to ensure that the fibers from the broken edge do not protrude past the edge-metal and prevent the connector from being fully seated on the edge of the board.

Q. How do the QXX-EM-GP (guide post option) footprint layouts differ than the standard QXX-EM footprint layouts?
A. The board alignment feature is thicker and spaced differently than the standard QXX-EM footprint. Otherwise, all other dimensions apply.

Q. Where do the tightest tolerances on the footprint layout belong?
A. The tightest board tolerances are located from the center of the alignment feature to the center of the first position pad, pad-to-pad PCB thickness, the width of the alignment feature and alignment of signal pads on top of board relative to those on the bottom.

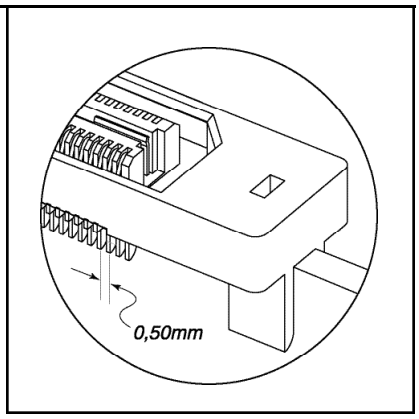
Q. What is Edge Metal?
A. *The plated edge that the ground plane solders to.*

These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

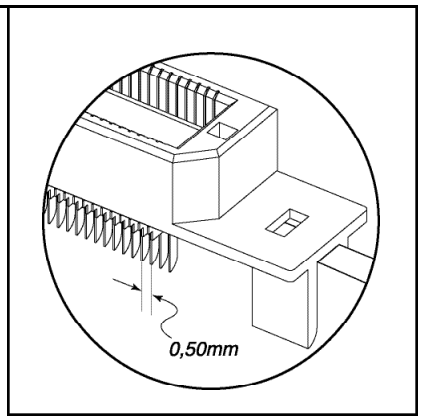
RECOMMENDED PCB FOOTPRINTS/STENCILS (PDF FORMAT):

0,50mm

QSH-EM
(Female)

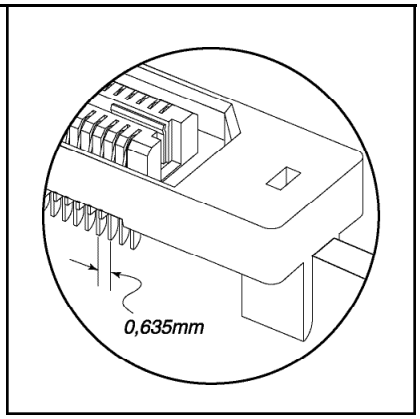


QTH-EM
(Male)

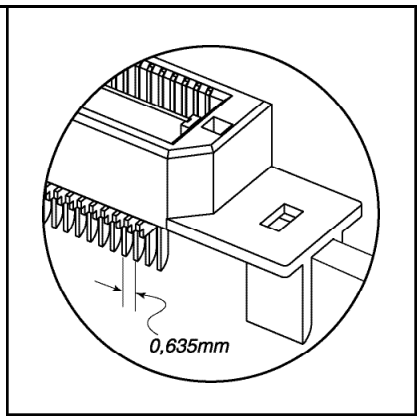


0,635mm

QSS-EM
(Female)

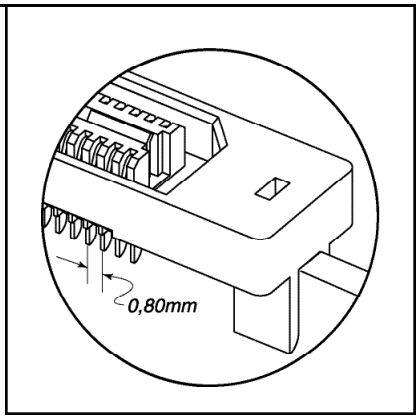


QTS-EM
(Male)

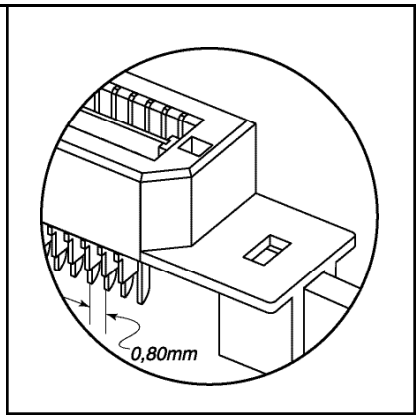


0,80mm

QSE-EM
(Female)



QTE-EM
(Male)



These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.

SAMTEC USA
P.O. Box 1147
New Albany, IN 47151-1147 USA
1-800-SAMTEC-9 (USA & Canada)
Tel:812-944-6733 • Fax:812-948-5047
E-mail: info@samtec.com

SAMTEC UNITED KINGDOM
117 Deerdykes View
Westfield, Cumbernauld
Scotland G68 9HN
Tel: 01236 739292 • Fax:01236 727113
E-mail:scotland@samtec.com

SAMTEC GERMANY
Frühlingstr. 14
82110 Germering • Germany
0800-SAMTEC9 (0800 / 72 68 329) Germany only
Tel:+49 (0) 89 / 89460-0 • Fax:+49 (0) 89 / 89460-299
E-mail: germany@samtec.com

SAMTEC FRANCE
4, allée Emile Reynaud
BUROPARC TORCY
77200 TORCY • France
Tél:01 60 95 06 60 • Fax: 01 60 95 06 61
E-mail: samtecfrance@samtec.com

SAMTEC ITALY
Via Colleoni 25
Centro Direzionale Colleoni
Palazzo Pegaso Ingresso 3
20041 Agrate Brianza-Milano • Italy
Tel: 39 039 6890337 • Fax: 39 039 6890315
E-mail:dbolzo@tin.it

SAMTEC ASIA PACIFIC
1 Kallang Sector #05-01
Kolam Ayer Industrial Park
Singapore 349276
Tel:65 6745-5955 • Fax:65 6841-1502
E-mail: samtec@singnet.com.sg

SAMTEC JAPAN
Aiku Bldg.3F
5-5-1 Kokuryo-cho
Chofu-city, Tokyo 182-0022 Japan
Tel:81-424-42-8318 • Fax:81-424-42-8319
E-mail: rhayashi@winter.try-net.or.jp

SAMTEC CHINA
East Ocean Centre, Suite 5E1
618 Yan An Road East
Shanghai, China 200001
Tel: 86-21-5385-4089 • Fax:86-21-5385-4047
E-mail: samtec.china@samtec.com

SAMTEC TAIWAN
4F, NO. 95, Sec.3, Roosevelt Rd.
Taipei 106
Taiwan ROC
Tel: 886 2 23646077 • Fax: 886 2 23679531
E-mail:samtectaiwan@samtec.com.tw

SAMTEC HONG KONG
Unit 10, 13th Floor, Block A, Veristrong Industrial Centre
34-36 Au Pu Wan Street
Fo Tan, Shatin, N.T.
Hong Kong
Tel: 852-26904858 • Fax:852-26904842
E-mail:samtechk@netvigator.com

SAMTEC ON-LINE
Internet:www.samtec.com

These guidelines and suggestions should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards and in your process to guarantee optimum results.