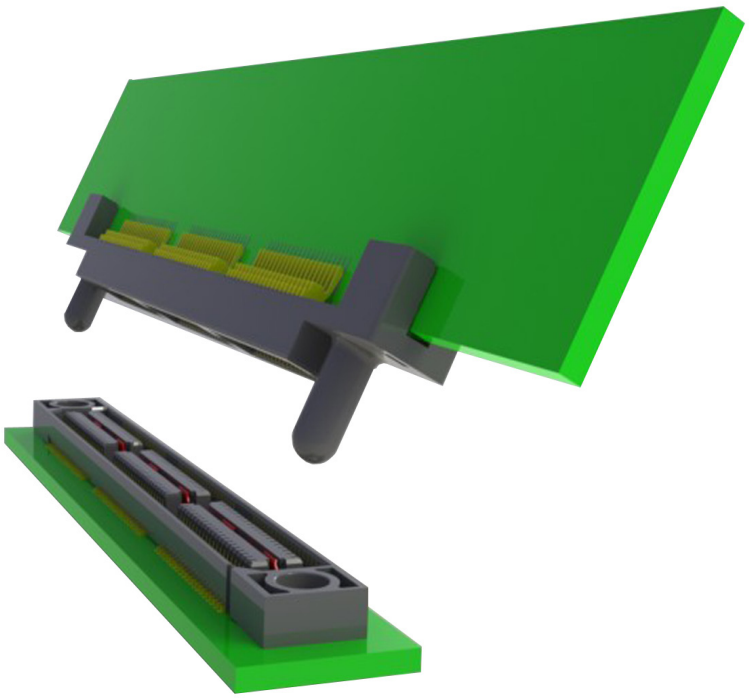
EDGE MOUNT Q-STRIP™/Q-PAIRS™ INTERCONNECTS APPLICATION OVERVIEW

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 printed circuit 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



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

PROCESSING OVERVIEW

The following guidelines were developed for Samtec's Q-Strip™ and Q-Pairs™ interconnect systems on .5mm, .635mm, and .8mm centerlines. Specifically, these guidelines apply to Samtec's QTH/QSH, QTS/QSS, and QTE/QSE series connectors with edge mount leads on .062" thick printed circuit boards (PCBs).

The micro pitch signal leads and 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.

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 achieve optimum results.

PROCESSING GUIDELINES

General Processing Equipment and Conditions:

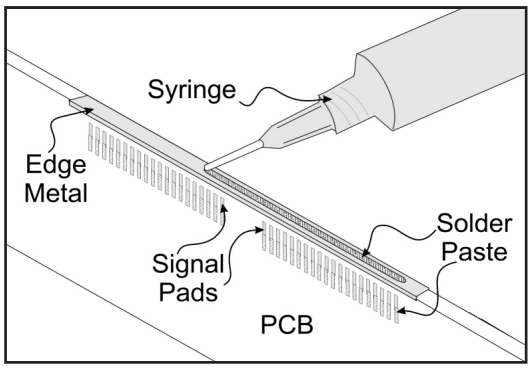
- .006" (0,15mm) stencil is recommended.
- Nominal PCB thickness is .062" (1,57mm) and the thickness tolerance shall be held as tightly as possible.
- Typical SMT process boundaries and oven speed/temperature settings may be used.
- An inert (nitrogen) reflow atmosphere is recommended for lead free applications.
- Connectors are manually placed on the PCB.
- Samtec's recommended footprint and stencil designs should be closely followed (see page 4 for links).

Procedure:

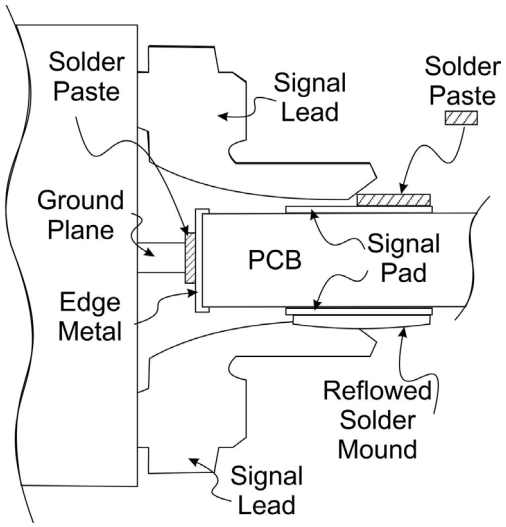
1. Apply solder paste to side one of the PCB, ***including*** the pads for the Q-Strip™ or Q-Pairs™ edge mount interconnect. ***Do not*** deposit solder on the edge of the PCB (edge metal).
2. Populate side one of the PCB, ***excluding*** the Q-Strip™ or Q-Pairs™ edge mount interconnect.
3. Reflow side one.
4. Flip the PCB and apply solder paste to side two, ***including*** the pads for the Q-Strip™ or Q-Pairs™ edge mount interconnect.
5. Apply tacky flux to the reflowed solder mounds on side one of the PCB.

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

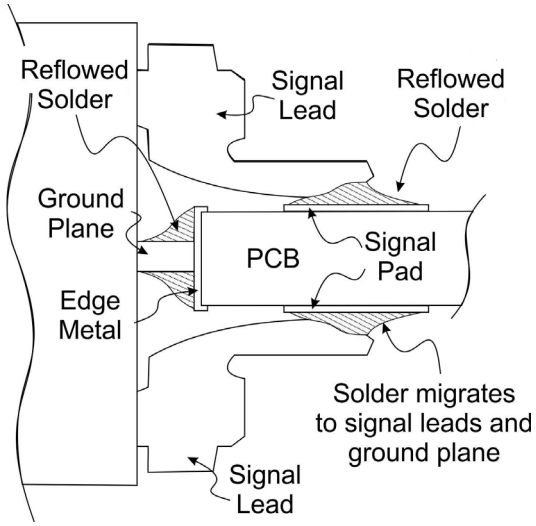
6. Apply solder paste to the edge of the PCB (edge metal) using a syringe or small putty knife:



7. Place the Q-Strip™ or Q-Pairs™ edge mount interconnect on the PCB along with the other side two components.



8. Reflow side two.



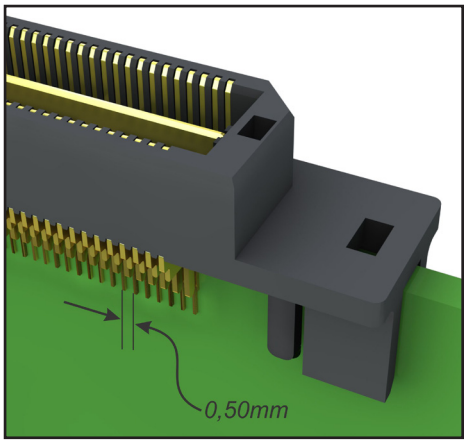
These guidelines should not be considered design requirements for all applications. Samtec recommends testing interconnects on your boards in your process to guarantee optimum results.

Links to Recommended PCB Footprints and Stencils
(PDF Format - Click on Picture)

0,50mm

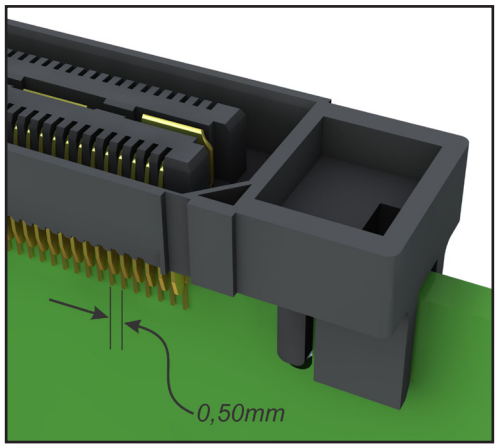
QTH-EM

(Male)



QSH-EM

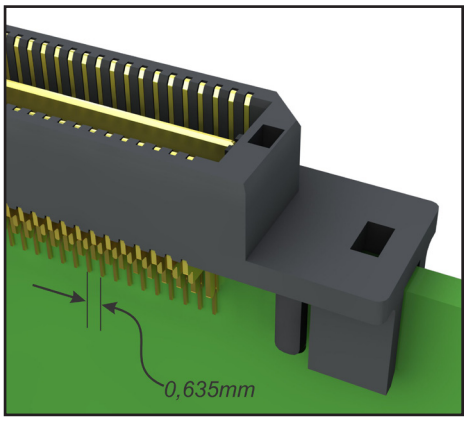
(Female)



0,635mm

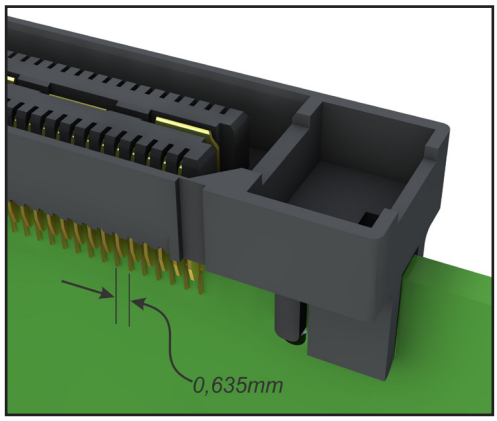
QTS-EM

(Male)



QSS-EM

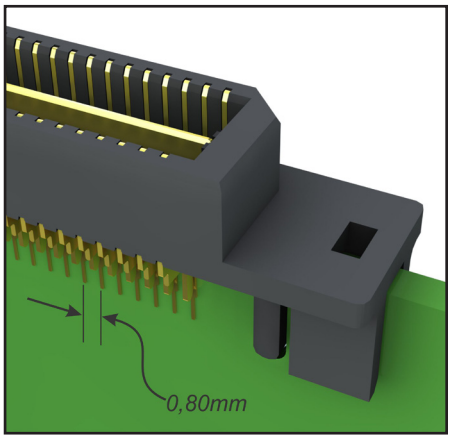
(Female)



0,80mm

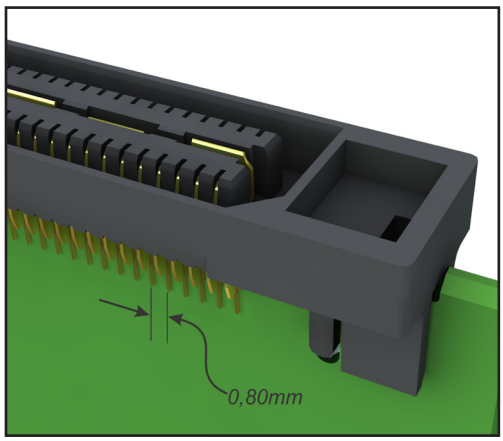
QTE-EM

(Male)



QSE-EM

(Female)



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