

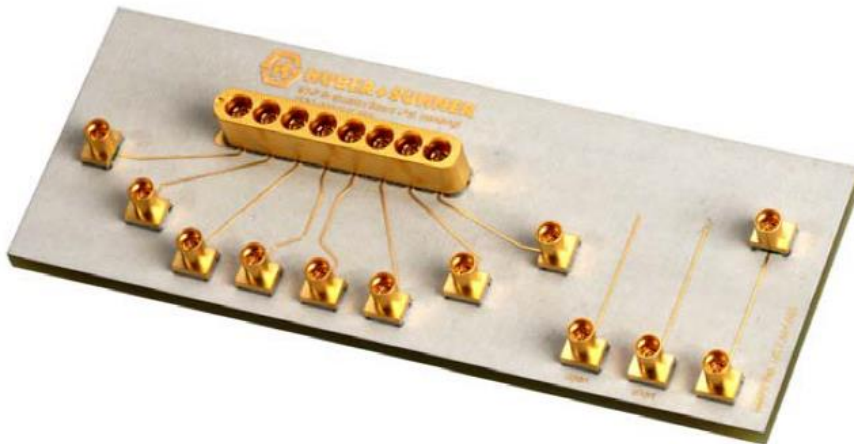
APPLICATION NOTE

HUBER+SUHNER AG
RF Division
R&D RF Coaxial Components
Product Management
2016

MMPX and MXP PCB Connector Assembling

Table of Contents

1	INTRODUCTION.....	1
2	PATTERN	2
3	FOOTPRINT	2
4	SOLDER STOP MASK.....	3
5	VIAS.....	3
6	PLATING	3
7	SOLDER PRINTING AND SOLDER STENCIL	3
8	COMPONENT PLACING	4
9	SOLDERING	4

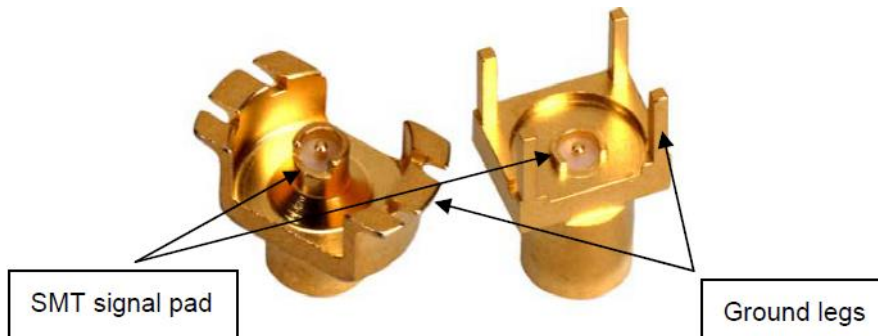


1 INTRODUCTION

MMPX and MXP are high performance PCB connectors. HUBER+SUHNER has developed certain design features to facilitate assembling of the connectors and offers engineering support for footprint optimization. To achieve the full rated electrical performance, the assembling process specifications have to be strictly followed.

2 PATTERN

MMPX and MXP are designed in SMT for RF performance reasons. However versions with ground legs are also available to improve mechanical stability and easy placement. Signal performance is identical for both designs.

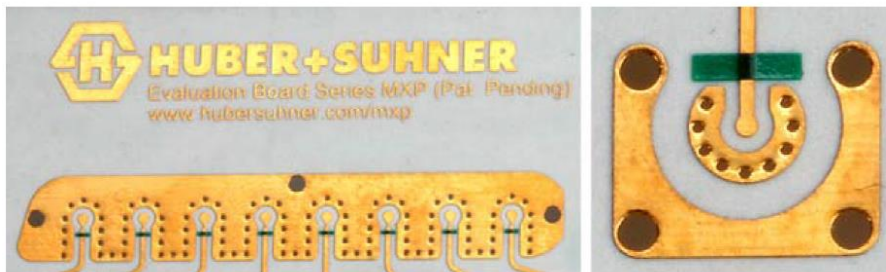


3 FOOTPRINT

A suitable footprint for components on PCBs is very important for a reliable production. For RF signals, matched impedance footprints are mandatory. Unfortunately such footprints depend on the base material. Relative dielectric constant, layer thickness and, in case of strip line, configuration of the complete PCB stack-up, all play a role

Typically such footprints are customer specific and the design requires electro-magnetic 3D field simulation.

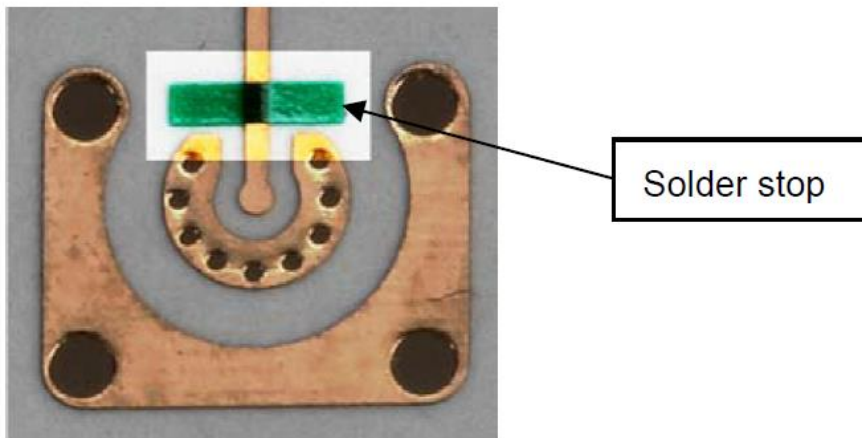
For PCB production it is essential to order impedance controlled PCBs and to choose a manufacturer with measurement capabilities and sufficient experience.



4 SOLDER STOP MASK

Solder mask is used to limit solder to pads and help to prevent short circuits. The drawback of solder mask is its high tolerance of dielectric constant and thickness. This influences the impedance of transmission lines. In any case, solder mask has to be specified and included in an impedance calculation.

Impedance matching can be improved if only a small spot of solder mask is placed next to the pad. This prevents solder from flowing away and has a negligible impact on impedance.



5 VIAS

HUBER+SUHNER recommends the usage of filled vias which prevents the solder paste from flowing away into the vias.

6 PLATING

Gold plating is excellent for soldering. Special care must be taken with thick gold layers, as gold will diffuse into solder and make it brittle, with concentrations above 3%.

Silver plating of traces can reduce the losses and improves signal integrity, but requires an additional processing step during manufacturing.

7 SOLDER PRINTING AND SOLDER STENCIL

HUBER+SUHNER has developed and tested an optimized solder stencil for reliable manufacturing and RF performance.

Correct printing requires a solder grain size not larger than the specified size. In case of MMPX and MXP type 3 is required. Ground legs are pin-in-paste compatible. Compared to SMT solder printing, pin-in-paste solder printing is not controlled by the stencil thickness but the printing parameters.

Ensure that a sufficient amount of solder is dispensed in to the hole, bearing in mind that solder paste contains only 50% solder. A relevant parameters is the wiper angle, where a smaller angle increases pressure.

Generally a solder roll of 25 mm is recommended. A closed printer head is recommended and helps solving many problems with pin-in-paste solder printing.

8 COMPONENT PLACING

MMPX and MXP are full pick and place compatible. All connectors in those families are supplied in tape and reel packaging. Where necessary a cap provides a flat area for the vacuum nozzle. Caps are design to remain on the connector while soldering. Moreover they can later be used to protect the interface from dirt.

For low volume production, the connector can be placed by hand. Any connector types with ground legs are preferred for manual placing as the legs help positioning the connector on the PCB.

The connectors are qualified for Unlimited Floor Live (Level 1) according JSTD 020D ed. 2007.

9 SOLDERING

Soldering of the connector requires industrial reflow equipment or at least a good rework station. A soldering iron or hot air pencils are not suitable.

The connectors were qualified as Not Process Sensitive (W0 and R0) according J-STD 075 ed. 2008.

9.1 Reflow

A hot air or vapour phase reflow is required to solder the components. MXP connectors have higher mass and therefore need more energy for heating up. But these connectors are tested for hot air reflow soldering and do not cause any problems when state-of-the-art equipment is used.

9.2 Wave Soldering

Ground legs can be soldered with the wave soldering technique. The legs should protrude 1.5 mm out of the PCB. For thicker PCBs, consider using pin-in-paste technology. For very small pitch components on the bottom side, consider selective wave soldering or pin-in-paste.

9.3 Pin-in-paste

For this technique a solder deposit is placed in the via hole to solder the legs. Make sure legs do not protrude more than 1.5 mm out of the PCB. In principle, legs do not need to protrude out of the PCB, but if so, then visual inspection has to be adapted, to not check for a meniscus and legs.