Surface-Mount Hybrid Transmitter & Receiver Handling AN 34

Introduction

RF Monolithics, Inc. (Murata) has developed new and innovative surface-mount (SM) pack-ages for its transmitter and receiver products, the *HX* and *RX* series. These packages meet the size / performance needs of wireless remote-control, security, and data-link ap-plications.

This note has been developed for our customers to assist in choosing appropriate end ap-plications and to make recommendations in the handling of these hybrid parts during the manufacturing process. These considerations are typical to those in use in the electronics industry for SM ceramic packages.

Background

The transmitter and receiver hybrids utilize SAW technology, which is the key to the small size as well as the high performance. SAW devices require a dry and clean environment, free of all corrosive gases. Proper operation is critically dependent upon the dielectric constants and mass loadings at the surface of the SAW. Therefore, a commercially viable SAW device must be hermetically sealed within a package that is filled with a low mois-ture, non-oxidizing, and non-corrosive gas. Dry nitrogen is the gas used for this purpose. A high degree of hermeticity and cleanliness must be maintained to insure the minimizing of outgassing from harmful chemicals and adhesives used in the manufacturing process. This hermeticity standard is required to meet military standards of MIL-STD-883, Method 1014. Murata applies this standard to all SAW components, including those products in-tended for low-cost consumer applications.

In the past, commercial parts have been supplied in metal cases with lids welded in place. The highest volume and hence lowest cost devices have been supplied in the 3-lead TO-39 case. These package styles utilize industry standard resistance welding techniques to provide a hermetic seal. Commercially available, industry standard, surface mount pack-ages suitable for SAW devices are used for mid to high end applications. These are typ-ically seam welded. Commercially available surface mount packages are prohibitively expensive for high volume components destined for cost sensitive consumer products.

Naturally, new and innovative packaging technology often requires new and often unique board design, handling, assembly, soldering, inspection and environmental consider-ations. Many of these considerations are similar to those already required in the SM in-dustry by more conventional SM component (capacitor, resistor, etc.) manufacturers. However, some requirements and precautions are unique to these new packages. The following information is intended as an aid to Murata customers in applying these compo-nents as efficiently and reliably as possible.

Surface-Mount Hybrid Transmitter & Receiver Handling AN 34

Basic Package Design

This note specifically applies to the new SM packages developed by Murata for the hybrid transmitters and receivers. Materials, metallization, assembly process, sealing, etc., are identical for both of these packages. Detailed package drawings are shown in our latest Murata Product Data Book.

The SM-4 package is a leadless chip carrier (LCC) containing the hybrid transmitter (Murata's HX series product). This active, 4-terminal device requires specific orientation. Terminal #1 is identified by a metallized "dot" on the bottom side of the substrate and a rectangular ink "dot" on the lid. The nominal footprint is approximately 96 mm^2 (0.149 in^2). Terminal width is 1.27 mm (0.050 in) with standard spacing of 5.08 mm (0.200 in).

The RX series of ASH receiver is provided in the 10-terminal, SM-10 LCC. This extremely compact and low current receiver is oriented by a metallized "dot" on the bottom side of the substrate and a rectangular ink "dot" on the lid to denote terminal #1. The terminal width is 1.27 mm (0.050 in) and are spaced on standard 2.54 mm (0.100 in) centers.

The ceramic base material for these packages is black alumina. The aluminum trioxide (Al_2O_3) content of this co-fired ceramic is a minimum of 90%. The metallic plating consists of electroless gold (Au), between 40 and 60 microinches thick, over a minimum of 50 microinches electroless nickel (Ni), over a base metal. This metallization must withstand 390°C in a dry nitrogen (N_2) atmosphere for 20 minutes without showing discoloration and must meet MIL-STD-883, Method 2003 solderability requirements.

The mechanical attachment and hermeticity between lid and substrate are provided by a glass frit seal. This "glass" is actually a composite material designed specifically for use in sealing alumina ceramic packages. The softening point of this glass is at 350°C and the transformation point is 260°C. (The transformation point is essentially a sudden change in the coefficient of thermal expansion and represents an increase in the sensitivity of the glass seal to damage from external mechanical stress.) This suggests that all post seal processes (i.e. soldering) should remain well below 260°C to ensure maximum integrity and reliability of this seal. A maximum of 250°C is recommended. See the Recommended Soldering Profile Chart that accompanies this Application Note for an overall representation of soldering specifications.

Package Handling and Environment

These new SM packages by Murata use industry standard alumina ceramics. The glass frit seal, a process that provides a very effective glass seal, can be damaged if handling pre-cautions are not observed. This section addresses handling precautions necessary for the reliability of this glass frit seal. Seal damage may result in a loss of hermeticity or total lid failure.

A loss of hermeticity means that the hermetic seal no longer meets MIL-STD-883, Method 1014 requirements discussed in previous paragraphs. A loss of hermeticity will cause accelerated frequency aging or a complete lid failure.

The following are a few easy precautions to take when handling these parts to avoid glass frit seal damage.

- Make sure the Pick-and-Place machines and alignment tools are properly adjusted to avoid excess impact on the glass seal. Pick-and-Place machines that require full face contact can overstress many parts including these.
- Excess printed circuit board flexing during board-break steps or during snap-fit steps causes severe strain on any surface mount component solder joint. The same flexing may also cause glass seal damage to the HX and RX parts in addition to solder joint damage.
- When setting up your soldering process, follow the Recommended Soldering Profile supplied with this application bulletin (Fig. 1). Maximum soldering temperature along with ramp up rate and cool down rate are important variables to consider when adjusting for your solder flow processes. Note: When hand soldering, as in engineering prototype assembly, care should be taken to avoid excess mechanical stressing on the parts. This, too, can result in seal damage.

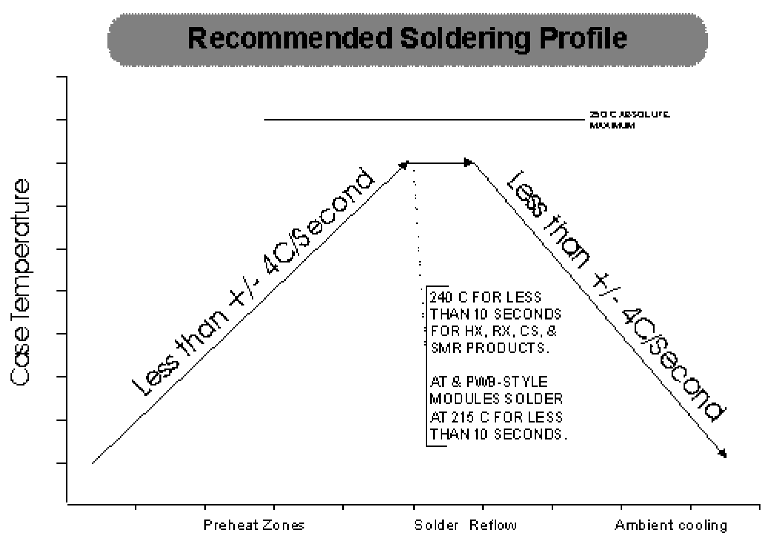


Figure 1

These packages are rugged with regard to constant forces to the lid in the normal operating temperature range. Lateral impact to the lid, especially those impacts that are directly to the glass seal, may cause seal damage. Damage is highly unlikely with normal handling that is appropriate for the industry standard ceramic glass SM packages. Further, production quantities of these devices are shipped in industry standard Tape-and-Reel which insulates the package from impacts during normal shipping and handling.

Damage during Pick-and-Place is nearly always caused by improper machine setup. The machine jaws should grasp the lid or base, not the glass frit seal. Centering and placement forces are comparable to industry standards for other standard ceramic SM components (typically 300g, max).

When severe testing and handling of boards cannot be avoided, potting the entire assembly (exclusive of the antenna) or epoxy sealing the perimeter of the HX and RX is a strategy than can be used to minimize risk of failure due to seal damage.

Recommended PCB Layout for Optimum Performance

Proper PCB layout will result in optimum performance for both the HX and the RX products. Please refer to the attached drawings for layout techniques for both.

- **The HX Layout**
  Refer to the layout drawing (Fig. 2) for the HX transmitter. The illustration shows pad spacing as being identical to the HX. It is permissible, as in Figure 1, to extend the pads beyond the body of the HX. This will allow for easier inspection of the solder joint. This not mandatory, however.

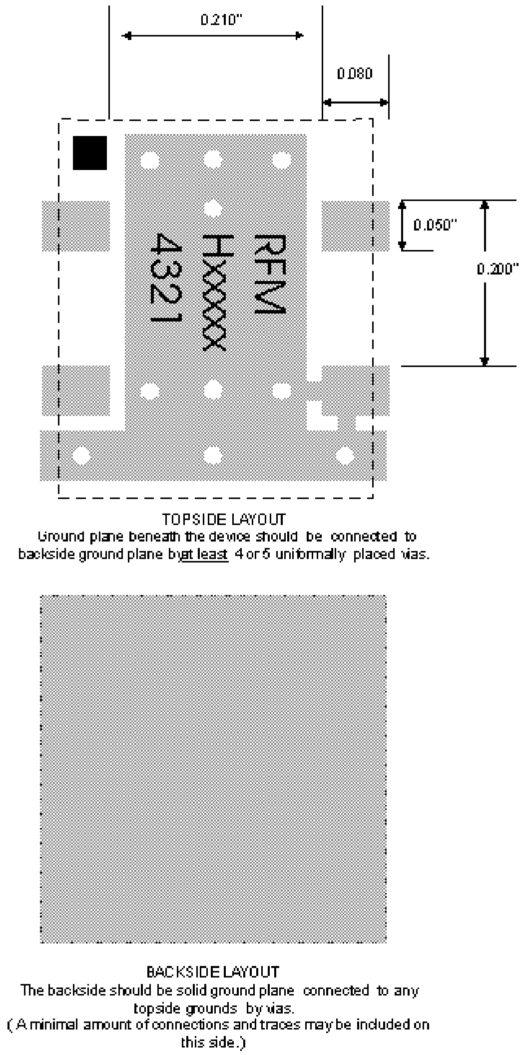
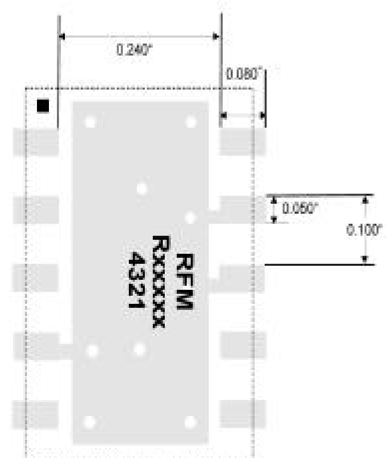


Figure 2

The ground plane, directly beneath the HX transmitter, is necessary for proper RF grounding and harmonic and spurious suppression. The ground pad (pin 3) should connect directly to this ground plane. Feed-through vias should connect this topside ground plane to a larger ground plane on the backside of the PCB. Several vias should be incorporated here; at least 3 to 5 vias are suggested at uniform intervals.

Inductor pads on the V_{CC} and Modulation input lines are often included to help suppress any tendency the design may exhibit toward radiating RF from these lines. This, too, will aid in passing any regulatory testing of the final design. Ferrite beads around a short piece of wire can be used in place of inductors on these pads, if preferred. Whether an inductor or a ferrite bead is used, these should be connected as close to the associated HX pins (pads) as possible to avoid any unnecessary etch length that could radiate RF energy.

- **The RX Layout**
  Refer to the layout drawing (Fig. 3) for the RX receiver. These pads match the spacing of the pads on the RX. As with the HX, the pads may extend out beyond the body of the receiver hybrid. This allows the easier solder joint inspection. Once again ground plane should be directly beneath the parts on the topside and this ground plane should be directly connected to a backside ground plane by numerous vias. Each RX ground pin should be connected directly to the topside ground plane.



TOPSIDE LAYOUT
Ground plane beneath the device should be connected to backside ground plane by at least 4 or 5 uniformally placed vias.

BACKSIDE LAYOUT
The backside should be solid ground plane connected to any topside grounds by vias
(A minimal amount of connections and traces may be included on this side.)

Figure 3

Bypass capacitors on the V_{CC} pin (pin #1) and V_{ref} pin (pin #6) should be included on any layout. These parts, like the inductors on the HX, should be positioned as close as possible to the actual RX pads.

- **Antenna Considerations**
  Both the HX and the RX require a 50Ω impedance match at the antenna pin. Since the antenna type will be determined by the system design requirements, it would probably be prudent to allow for tuning or matching components between the antenna and the RF Output for the HX and the RF Input for the RX. These tuning components may include series as well as shunt components. Distance between the RF pin of the device and the tuning components as well as the antenna, should be kept to a minimum since etch adds inductance that may not be desirable. If it is necessary to have the antenna at a distance from the output pin, then a microstrip (50Ω) should be used to make the connection. Since the antenna system can "make or break" any RF design, careful planning and design techniques should be utilized to optimize this part of the design.

- **Rework of HX and RX Devices**
  In the event that an HX or RX device needs to be removed from a circuit board, the following procedure should be observed:

  CAUTION: Do not try to remove the device with a soldering iron. This could result in damage to the HX/RX device.

  Select a hot air rework station similar to the Pace ThermolFlo (PN 8007-0296) which has the capability of maintaining a soldering profile as specified in the application notes. Set the air blower speed, temperature and cycle time controls to the required settings to optimize performance. Install the appropriate QuickFit Nozzle (PN 4028-4001 for an 8 pin device or 4028-4002 for a 14-16 pin device) onto the ThermolFlo handpiece. Place PCB assembly onto the work holder and apply flux to the component leads (optional). Position the nozzle over the component to be removed. Adjust the Vacuum Pik to the proper height so the vacuum cup is at the bottom edge of the ThermolFlo nozzle. Lower the nozzle over the component to a point where the bottom of the nozzle is 0.030-0.050" above the PCB. Commence reflow cycle and observe solder melt at all leads. After complete solder reflow, press the vacuum switch and raise the nozzle (with the component attached to the vacuum cup) from the PCB. Allow your board to cool prior to removing from the work holder.

  Alternative method:
  For single sided boards, a hot plate may be used. Allow the board to heat until the solder begins to reflow and remove the component with a pair of tweezers or vacuum pen. Allow the board to cool after removing it from the hot plate.