AMMP-XXXX
Production Assembly Process

Application Note 5386

Description
This application note describes and illustrates the attachment and detachment of Avago's 5x5 mm packaged products in a PCB (soft board) environment. The AMMP package has a solid copper filled cavity to allow optimum thermal transfer of the packaged device to the PCB and associated heat sink interface.

The AMMP package takes into full account the parasitics involved with routing a high frequency signal up and through a ground plane transition; therefore, these guidelines also illustrate measures that will keep the RF signal interface at an optimum. Figure 1 illustrates an AMMP package assembly. Figure 2 is a cross-section view.

Package Features
- Surface Mount Package (5x5 x 1.25 mm)
- Tape and reel capability for chip shooter pick-and-place applications.

Figure 1. The AMMP-XXXX DC-40GHz Package

Figure 2. Cross-section View of AMMP-XXXX Package
Package Dimension Tolerances

Stencil Type
A stainless steel stencil with trapezoidal walls is recommended. If possible, electropolishing for smooth walls will reduce surface friction and promote good paste release. The trapezoidal section aperture (TSA) design also promotes precision solder deposits to assist in component placement and alignment. The resulting, reflowed solder joints on the perimeter I/O lands should measure approximately 50 µm to 75µm (2 to 3 mils) standoff height. Figure 3 is a drawing for a stencil pattern that optimizes the amount of solder paste that maximizes contact coverage and minimizes thickness, which would degrade RF signal performance.

![Figure 3. Solder Stencil Dimensions](image)

Solder Paste
Recommended solder paste is SolderPlus 62NCLR-A/Sn62 Pb36 Ag2 (Engineered Fluid Dispensing). Details can be obtained from http://www.efd-inc.com

If the vias on the PCB are not silver filled, taping the backside of the PCB with thermal tape, prior to dispensing the solder paste, is recommended.

This will help containment of the solder in the vias which is critical for proper thermal transfer from the package to the PCB and heat sink. Remove thermal tape after the cool down period of the solder process.

If lead-free solder is required, a “no clean” Type 3 paste per ANSI/J-STD-005 is recommended. Nitrogen purge is also recommended during reflow.

For automated assembly, solder paste should be screen printed before reflow. Figure 4 illustrates the solder deposit pattern superimposed on the PCB.

![Figure 4. Epoxy Pattern on PCB](image)

Manual Solder Dispersion Guidelines
For manual dispersion of solder paste on the PCB, it is critical to distribute the solder paste evenly in the same pattern as shown in Figure 4 above. The amount should be distributed in an even pattern at a thickness of approximately 0.1 mm (0.004”), because when solder is unevenly distributed it may result in voids underneath the package or non-contact on parts of the package bottom. Too much solder paste may result in bridging of the contact pads. Figure 5 illustrates possible conditions that may occur with poor distribution of solder paste.

![Figure 5. Solder Dispensation Illustrations](image)
PCB Land Pattern and Material

The PCB material and mounting pattern, as defined in the data sheet, optimizes RF performance and is strongly recommended. An electronic drawing of the land pattern is available upon request from Avago Sales and Application Engineering. Figure 6 is a drawing of the land pattern. Material is Rogers RO4350 with thickness of 0.25 mm (0.010”).

For products with approximately 2 W dissipation or greater, silver filled vias are highly recommended.

![Figure 6. AMMP-XXXX Land Pattern](image)

Solder Reflow

Reliable assembly of surface mount components is a complex process that involves many material, processes, and equipment factors, including: method of heating, circuit board material, conductor thickness, pattern, type of solder paste and solder alloy, thermal conductivity, and thermal loading and mass of components. Reflow temperature settings need to be determined by the user based on these considerations. Also, moisture sensitivity level (MSL) 2A has been qualified for this device. The MSL 2A conditions must not be exceeded.

After ramping up from room temperature, the circuit board with components held in place with solder paste, passes through one or more preheat zones. The preheat zones increase the temperature of the board and components to prevent thermal shock and evaporation of solvents from the solder paste. The reflow zone briefly elevates the temperature sufficiently to produce a reflow of the solder.

Profile for Solder Reflow

The reflow profile investigated and recommended by Avago for this product is based on JEDEC/IPC standard J-STD-020 revision C. This device has been qualified to withstand a maximum of three cycles of solder reflow according to the conditions of J-STD-020C, and this device has been qualified for moisture sensitivity level 2a. Figure 7 depicts this standard lead-free JEDEC/IPC profile. Table 1 lists the parameters and peak temperatures as indicated by JEDEC/IPC. The most recommended and most common reflow method is accomplished in a belt furnace using convection/IR heat transfer.

Table 1 shows the actual temperature range that should occur on the surface of a test board at or near a central solder joint. During this type of reflow soldering, the circuit board and solder joints tend to heat first. The components on the board are then heated by conduction. The circuit board, because it has a large surface area, absorbs thermal energy more efficiently, then distributes this heat to the components.

Do not use prolonged hot preheat due to excessive oxidation which can occur on the solder powder surface. The time at peak is not critical and usually not measured as it is very dependent upon the type of oven used. However, time over 217° C is critical and will determine the appearance of the solder joint after reflow. Longer reflow time may result in dull and gritty solder joint appearance and charring of flux residues. Time below 30 seconds may result in insufficient wetting and poor intermetallic formation. Too fast of a cooling rate could result in insufficient wetting and poor intermetallic formation.

As a general guideline, this package should be exposed to only the minimum lead-free process temperature and times necessary to achieve a uniform reflow of solder on the board. The rates of change of temperature for the ramp-up and cool-down zones are specified by J-STD-020C standard to be low enough to not cause deformation of the board or damage to components due to thermal shock. This profile allows a reflow temperature which is low enough to avoid damaging the internal circuitry during solder reflow operations provided the time of exposure at peak reflow temperature is not excessive.

Also refer to the AMMP-XXXX product data sheet for further information.
Table 1. Typical SMT Reflow Profile for Maximum Temperature = 260+0 / -5°C

<table>
<thead>
<tr>
<th>Profile Feature</th>
<th>Sn-Pb Solder</th>
<th>Pb-Free Solder</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average ramp-up rate (TL to TP)</td>
<td>3°C/sec max</td>
<td>3°C/sec max</td>
</tr>
<tr>
<td>Preheat</td>
<td></td>
<td></td>
</tr>
<tr>
<td>– Temperature Min (Tsmin)</td>
<td>100°C</td>
<td>150°C</td>
</tr>
<tr>
<td>– Temperature Max (Tsmax)</td>
<td>150°C</td>
<td>200°C</td>
</tr>
<tr>
<td>– Time (min to max) (ts)</td>
<td>60-120 sec</td>
<td>60-180 sec</td>
</tr>
<tr>
<td>Tsmax to TL</td>
<td></td>
<td></td>
</tr>
<tr>
<td>– Ramp-up Rate</td>
<td>3°C/sec max</td>
<td></td>
</tr>
<tr>
<td>Time maintained above:</td>
<td></td>
<td></td>
</tr>
<tr>
<td>– Temperature (TL)</td>
<td>183°C</td>
<td>217°C</td>
</tr>
<tr>
<td>– Time (TL)</td>
<td>60-150 sec</td>
<td>60-150 sec</td>
</tr>
<tr>
<td>Peak temperature (Tp)</td>
<td>240 +0/-5°C</td>
<td>260 +0/-5°C</td>
</tr>
<tr>
<td>Time within 5°C of actual Peak Temperature (tp)</td>
<td>10-30 sec</td>
<td>20-40 sec</td>
</tr>
<tr>
<td>Ramp-down Rate</td>
<td>6°C/sec max</td>
<td>6°C/sec max</td>
</tr>
<tr>
<td>Time 25°C to Peak Temperature</td>
<td>6 min max.</td>
<td>8 min max.</td>
</tr>
</tbody>
</table>

Figure 7. Standard J-STD-020C Lead-free Solder Reflow Profile.
Desoldering the AMMP Package

Desoldering can be done by hot-air, hot-tip contact and on a hot plate. An example of hot-plate desoldering is shown in Figure 8. An aluminum block, sized to fit under the 5x5 mm package area, is used to transfer the heat from the hot plate. The aluminum block centralizes the heat under the package and avoids reflow of surrounding components. When the solder has reflowed, the package should be removed by gripping it with tweezers from the bottom section. Figure 9 shows various blocks, a typical hot plate set to approximately 200° C and an AMMP PCB assembly. Figures 10 and 11 show the removal of the AMMP package using tweezers.

Figure 8. Hot Plate Desoldering

Figure 9. Typical Hot Plate and Components

Figure 10. AMMP on Aluminum Block

Figure 11. Removal of AMMP Package
Hot Air Desoldering

Hot Air Equipment

Figure 12. AMMP-XXXX Soldered on PCB

HAKKO FM202

Nozzle: A1130

Figure 13. PCB Secured for Access Underneath PCB

Figure 14 shows an illustration of blowing hot air underneath the demonstration board by using the hot air nozzle and at the same time using tweezers to grip the bottom side walls of the AMMP package.

Figure 14. Blowing Hot Air Underneath the Demonstration Board
Hot Tip Contact Desoldering

Hot Tip Equipment

Direct contact is made under the package surface area.

Lift package, with tweezers, from bottom side walls when solder has reflowed, as shown in Figure 14.

Keep direct contact under the package surface area to reflow solder.

Figure 15. Hot Tip Desoldering
Special Handling Notes for Automatic and Manual SMT Rework

- Package should be cooled down before any solution cleaning after IR reflow.
- Solder coverage should be evenly distributed across the bottom contact area to obtain good, reliable DC and RF contact.
- Alignment and good wetting of the RF In/Out pad to the PCB is critical to ensure optimum signal transmission.
- Handling the package along the edges with tweezers or auto placing with vacuum force from the top of the lid is recommended.
- DC probe or other sharp instrument contact is prohibited in the bottom center area outlined in Figure 16. DC probe contact or other sharp instrument contact is allowed in the area marked “A”.
- For volume production, the AMMP package can be treated as a standard surface mount component (ref. IPC/JEDEC J-STD-020C std. or equivalent) with a standard assembly process. (eg, stencil solder printing, standard pick and place, and solder reflow oven).

Figure 16. Prohibited Probing Area
RF Port Transitions

It is important to note that RF port hardware, such as SMA 2.92 mm or 2.4 mm connectors, will directly contribute loss and limit the AMMP performance within the hardware's bandwidth.

Table 2 is a list of recommended edge launch connectors, used in Avago's mmWave evaluation board modules. Other connectors with similar performance will also be suitable.

Table 2. Recommended Edge Launch Connectors

<table>
<thead>
<tr>
<th>Connector Type</th>
<th>Bandwidth</th>
<th>Manufacturer</th>
<th>Female Part Number</th>
<th>Male Part Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>SMA (screw on)</td>
<td>DC-27GHz</td>
<td>Southwest Microwave</td>
<td>292-06A-5</td>
<td>293-03A-5</td>
</tr>
<tr>
<td>SMA (solderable)</td>
<td>DC-26.5GHz</td>
<td>Johnson</td>
<td>142-0761-841-A</td>
<td></td>
</tr>
<tr>
<td>SMA (solderable)</td>
<td>DC-26.5GHz</td>
<td>Johnson</td>
<td>142-0761-851-A</td>
<td></td>
</tr>
<tr>
<td>SMA (solderable)</td>
<td>DC-26.5GHz</td>
<td>Johnson</td>
<td>142-0761-891-A</td>
<td></td>
</tr>
<tr>
<td>2.92 mm (screw on)</td>
<td>DC-40GHz</td>
<td>Southwest Microwave</td>
<td>1092-04A-5</td>
<td>1092-03A-5</td>
</tr>
<tr>
<td>2.4 mm (screw on)</td>
<td>DC-50GHz</td>
<td>Southwest Microwave</td>
<td>1492-03A-5</td>
<td>1493-03A-5</td>
</tr>
</tbody>
</table>

Manufacturer URL:
http://www.southwestmicrowave.com
http://www.EmersonNetworkPower.com

Bypass Capacitors

Bypass capacitors, as recommended at the DC pin outs on Avago AMMP data sheets, should be as close to the package as possible. Table 3 lists two typical values and their part numbers. Other capacitors similar in specification may be substituted.

Table 3. Bypass Capacitors

<table>
<thead>
<tr>
<th>Capacitor Type</th>
<th>Value</th>
<th>Manufacturer</th>
<th>Part Number</th>
</tr>
</thead>
<tbody>
<tr>
<td>SMT-0402</td>
<td>0.1 µF</td>
<td>AVX</td>
<td>AVX0402YG104ZAT2A</td>
</tr>
<tr>
<td>SMT-0805</td>
<td>&gt;1 µF (4.7 µF)</td>
<td>Phycomp</td>
<td>08052F475Z688OD</td>
</tr>
</tbody>
</table>