AH201/QFN Package Hand-Soldering Application Note

Summary:
This Application Note discusses a procedure to assist WJ customers in properly mounting the AH201 quad, flatpack, no-lead (QFN) package to printed circuit boards (PCB) using manual hand-soldering methods. While the example shown in this application note is with an AH201 device and application board, the basic procedure can also be applied to other WJ devices housed in similar QFN packages.

Introduction:
The AH201 is a 1-Watt driver amplifier offering high dynamic range in a low-cost, 6 x 6 mm, 10-pin surface mount package. Because of its excellent linearity (+47 dBm), P1dB performance (+30 dBm), and high gain performance (17 dB); the AH201 has many uses as a driver amplifier for many types of mobile infrastructure or broadband applications (50 – 2200 MHz).

Achieving these levels of performance requires a fair amount of DC dissipation for the device. The AH201 typically dissipates 11 V x 350 mA = 3.85 Watts, assuming that little or no DC energy is converted to RF energy. To dissipate that amount of heat, the device is placed in a QFN package, which takes up the same board space as a SOIC-8 package (including the leads). The QFN housing offers a convenient surface mount package capable of properly transferring the heat energy generated by the die to the backside of the package. More details are shown in the WJ Application Note "AH201 - Mounting Considerations for Medium Power Surface Mount RF Devices".

The QFN package is intended and ideally suited for high-volume, automated solder-reflow procedures typically used by contract manufactures. It is permissible, however, to hand-solder and manually mount the AH201 device onto a PCB board for prototype and engineering evaluations. The remainder of this application note details a proper method for performing this manual assembly procedure.

Procedure:
1. A properly laid-out PCB with an ample amount of via holes must be used to mount the AH201. The vias are used both to transfer heat during the manual assembly procedure and to also dissipate heat efficiently from the PCB during normal operation. Details for the recommended land pattern are shown in the AH201 Datasheet. The backside of the PCB must also be exposed and not contain solder mask in order to heat up the backside of PCB for this procedure. The AH201 PCB used for AH201 Sample Application Boards is used to investigate the procedure and is shown in Figure 1.

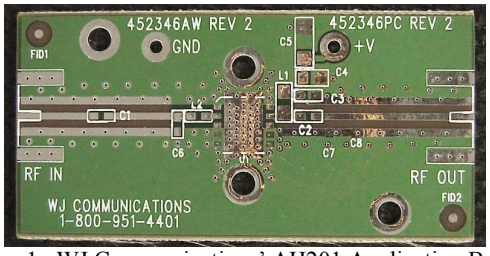

Figure 1. WJ Communications' AH201 Application Board.

2. The next step is to securely mount the PCB to a vice, clamp, or other rigid structure. It is best to only clamp a portion of the PCB in a manner similar to Figure 2, so that the user can access both the top and bottom sides simultaneously with a soldering iron.

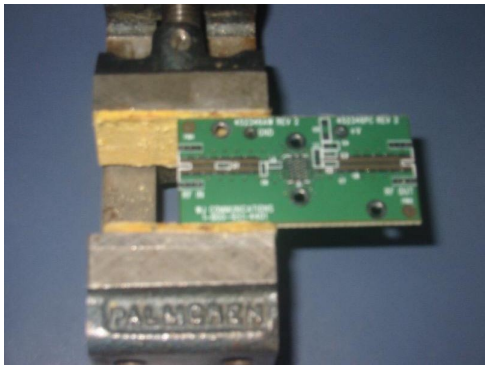
Figure 2. PCB shown mounted to a clamp.

3. As a preparatory step prior to installing the device, an ample amount of solder should be applied and left on all 10 device leads and on the large ground pad on the PCB as shown in Figure 3.

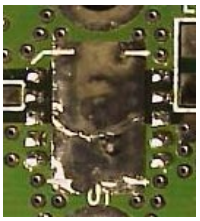

Figure 3. PCB shown with solder added to ground pad and leads.

4. Plenty of soldering flux should be added to the working area where the device is to be mounted. This assists in achieving good solder reflow during the device attach.

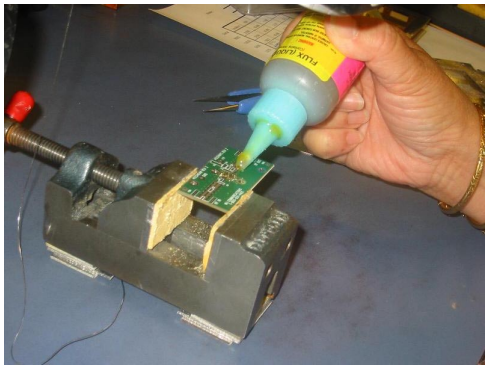
Figure 4. Application of extra flux to the device mounting area.

5. The device should now be placed and aligned on the area where it is to be mounted. Aligning the signal and ground leads visible on the edge of the device with the land pattern on the PCB can assist with the positioning of the device.

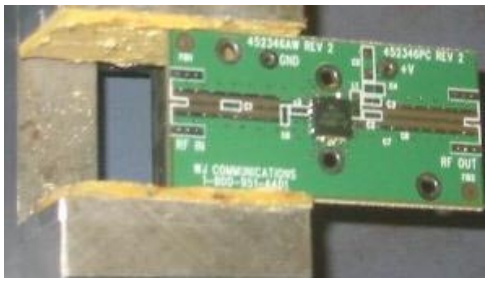

Figure 5. PCB shown with device lined up over land pattern.

6. A large-tipped soldering iron should be used to apply the amount of heat necessary for good solder reflow through the .062" thick PCB. A tip size of 0.2" x 0.3" (5 x 7.6 mm) is recommended. The soldering iron tip used for this exercise is a Metcal STTC-117 and is diagrammed in Figure 6, although exact tip type is dependent on the user's soldering iron.

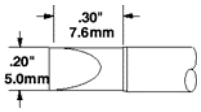

Figure 6. Diagram of a proper sized soldering tip.

7. The soldering iron should be applied to the backside of the PCB directly underneath the device as shown in Figure 7. The solder between the device and the PCB will start to melt and slowly attach the device.

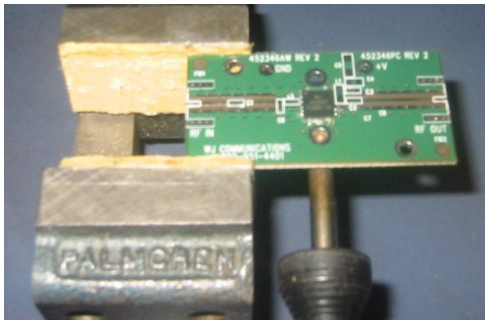

Figure 7. Heat is applied to the bottom of the PCB beneath the device.

8. While the solder is molten, the user should carefully align the device to the land pattern designated on the PCB for a proper attach as shown in Figure 8. It is normal to expect some solder to fill the vias underneath the device. As this happens, the solder should attach the device and make it flush with the PCB. Fillets may be seen on the exposed copper on the sides of the device to indicate a good attach.

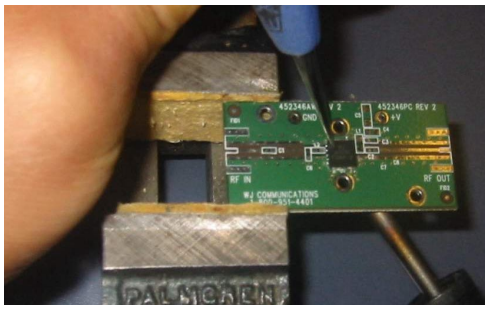

Figure 8. The device is aligned as the solder is molten.

9. To determine whether a good attach is made, several visual inspections can be made. Excess solder should be seen on the exposed copper on the PCB and on the device on all four sides as shown in Figure 9. As stated previously, solder fillets on the edges of the device indicate a good attach. This can be seen on a side view of the device in Figure 10. In addition, the user can check on all four sides to determine if the part is indeed flush with the PCB. A view of the backside will show that several vias will be filled in with solder as seen in Figure 11. This is normal and desirable from a heat transfer standpoint.

Figure 9. Angled top/side view of mounted device on the PCB.

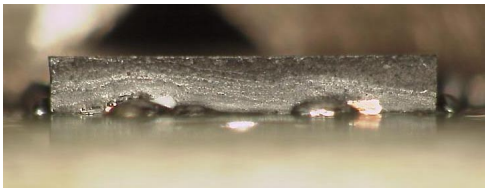

Figure 10. Side view of soldered device on the PCB.

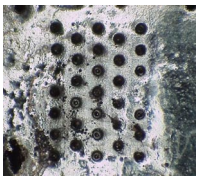

Figure 11. Backside view of the PCB underneath the device.

10. After confirmation of an acceptable attach, the excess flux and residue should be cleaned from the PCB with alcohol or other appropriate cleaning solvents. All other components can now be soldered to the PCB. The task for soldering the 6 x 6 mm QFN device is now completed.

Figure 12. PCB shown with properly soldered AH201 device.