



REFLOW PROCESS CONSIDERATION ON GLOBALTOP GPS/GNSS MODULE



Version History

Title: Reflow Process Consideration on GPS/GNSS modules

Subtitle: GNSS Module

Doc Type: Datasheet

Revision	Date	Author	Description
V0A	2013/03/08	Sidney	First Release



Table of Contents

1. Scope.....	4
2. Summary	5
3. Typical Structure of a GPS Antenna Module	7
4. Reflow	8
4.1 Reflowing Mirrored PCBA Boards	8
4.2 Reflowing Thin PCBA Boards	8
5. PCBA Handling	11
6. Module Rework.....	12
6.1 Rework on a Standalone Module	12
6.2 Rework on a Patch Antenna Module	15



1. Scope

This documents serves and provides general considerations as a reference to the reflow operator for running reflow on GlobalTop surface mount GPS/GNSS modules. Different SMT machines may have different characteristics. Different modules will, again, have different situations. Both will alter reflow condition. Cautions and suggestions are given in the table below.

SL3C –stand alone module and PA6C –patch antenna module are used in this document to help better describe use case and conditions discussed.

2. Summary

Caution items in the table offers quick reference guide for the reader. Details are provided in the following chapter.

Item No.	Item	Caution	Notes
1	Module preheating prior to reflow	<ol style="list-style-type: none"> For on tray package, tune heat to 60°C (±5° C) for 4hrs. For Tape&Reel package, tune heat to 60°C (±5° C) for 4hrs. 	<p>The heat tolerance of the provided tray is up to 100°C. Anything above, deformation will occur.</p> <p>Note: Make sure to allow the package to cool down to room temperature before handling to avoid deformation.</p>
2	<ol style="list-style-type: none"> For mirrored PCB board reflow For double-sided PCB board reflow 	<ol style="list-style-type: none"> Ensure that there is enough spacing in between the GPS module and the solder pallet to avoid contact. For a double side reflow, ensure that components on the host board are not in contact with any parts of the reflow machine or solder pallet during reflow and SMT process. 	<p>While the module is hot, parts on the GPS module may be shifted when come in contact with solder pallet causing external pressure.</p>
3	During SMT component placement process for GPS modules that are at the bottom side of the board.	During and between the subsequent reflow processes, operator must handle with caution with the pallet lifter that the active pin or tooling pins must not come in contact with SMT components causing damages.	<ol style="list-style-type: none"> Active pins may cause surface scratches and potential module component shifts. Operator must always examine the accuracy of the SMT component placement. Inform engineer for assistance for any placement issues.

Item No.	Item	Caution	Notes
4	The must-know about SMT Reflow	<p>1. During the reflow, deformation of the PCBA may occur. This will cause GPS module to shift, blown off, and etc.</p> <p>2. When the GPS module is positioned at the bottom during the second-reflow process, we recommend limiting the bottom heat under 200°C. Such operation should not, however, negatively affect the soldering process on the upper side of the board.</p> <p>3. Prior to the reflow process, operator must always examine SMT component placements are accurate and whether any components are misaligned or even reverse polarity.</p> <p>4. During SMT process for pilot production and mass production, logs must be created for each production made. Log is to be kept for a least a year is highly recommended for product issue tracking.</p>	<p>1. During the SMT process, when the defective rate is over 0.1%, stop the production immediately to prevent increase of defective rate making rework more difficult; and provide any necessary steps to such issue.</p> <p>2. The recommended peak heat for GlobalTop's GPS modules is anywhere between 243°C and 247°C with duration of no more than anything between 45~50 seconds.</p> <p>3. The third cycle reflow specifically for rework is not recommended.</p>
5	Rework for defects resulted from poor SMT placements	Extra caution must be taken when reworking on defect GPS modules not to apply pressure with any tooling on the patch antenna and/or Shielding cover of the GPS module as this may cause internal short circuits.	<i>Refer to the Structure of a GPS Antenna Module below.</i>

3. Typical Structure of a GPS Antenna Module

The GPS Antenna Module, for example, PA6C module consists of ceramic patch antenna, shielding cover, and PCBA in a top-to-bottom order. The PA6C module in this example has the Net Weight of 6 gram.

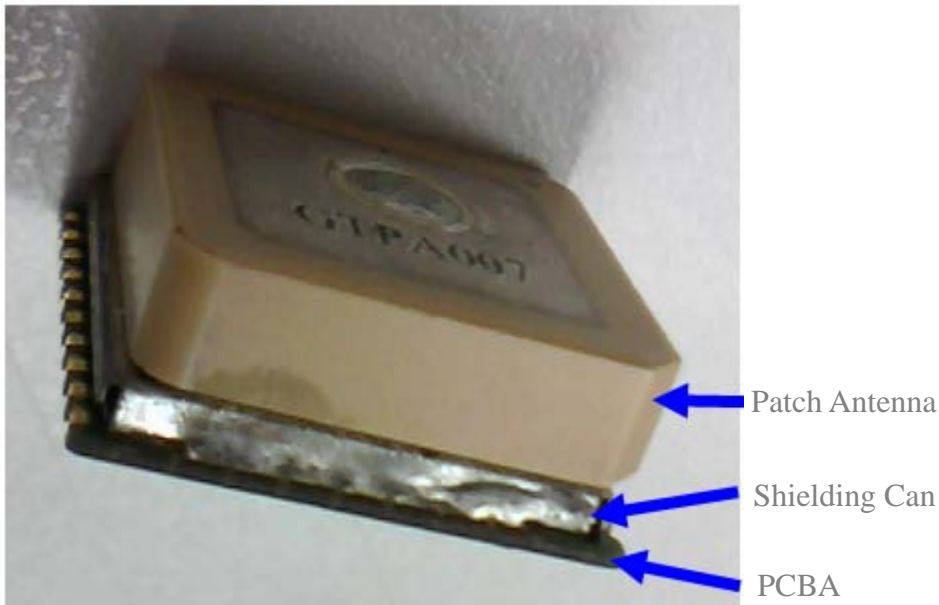
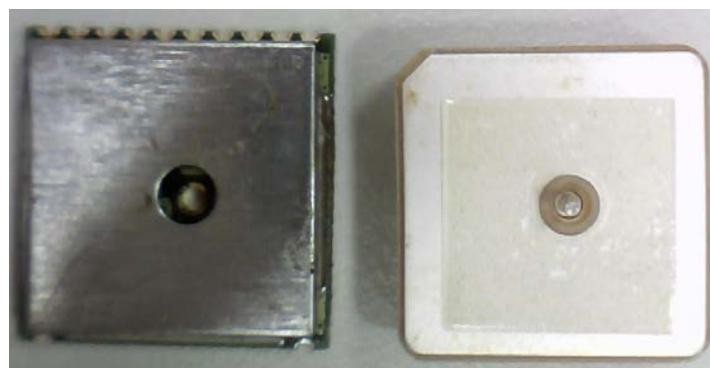


Figure below demonstrates the patch antenna being detached from the PC6C module leaving the shielding cover on top of the PCBA.



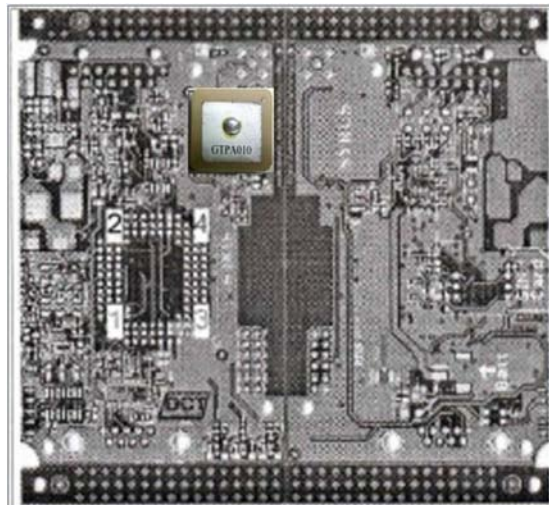
Shielding Can on PCBA

Patch Antenna

4. Reflow

4.1 Reflowing Mirrored PCBA Boards

1. During the second reflow process, GPS module placement shifts, moves or blown off at the bottom of the board were often occurred at the liquidous temperature range of the reflow profile. This can easily happen due to parts such as patch antenna and shielding cover are materials that retain heat.
2. We recommend limiting the bottom heat under 200°C at the cool-down of the reflow profiling or by turning the bottom heating zone off. Such operation should not, however, negatively affect the soldering process on the other side of the board.
3. Placement shifts would often happen to heavy modules such one that contains patch antenna. After completed the reflow process, allow the GPS module to be cooled to room temperature before handling. Care should be taken during the transportation of the PCBA not to cause internal parts shifts in the GPS module.



4.2 Reflowing Thin PCBA Boards

1. As rework for patch antenna modules are complex tasks, so prior to soldering reflow process, ensure that the placement and alignment of patch antenna module is properly done.
2. In a standard reflow configuration for thin PCBA, PCBA fixture or solder pallet is often utilized. When the GPS module is positioned at the bottom, ensure that the module is not touching the solder pallet otherwise the RF performance of the module can be affected.

3. After reflow, allow PCBA to be cooled to room temperature before handling. Smearing of solder paste is very likely to occur when handling PCBA while hot. Internal part shifts of the module are often not visible. The module alignment may still look perfect from the outside; noted that materials such as patch antenna retain heat.

Figure below shows proper SMT component placement/alignment of SMT module prior reflow process.

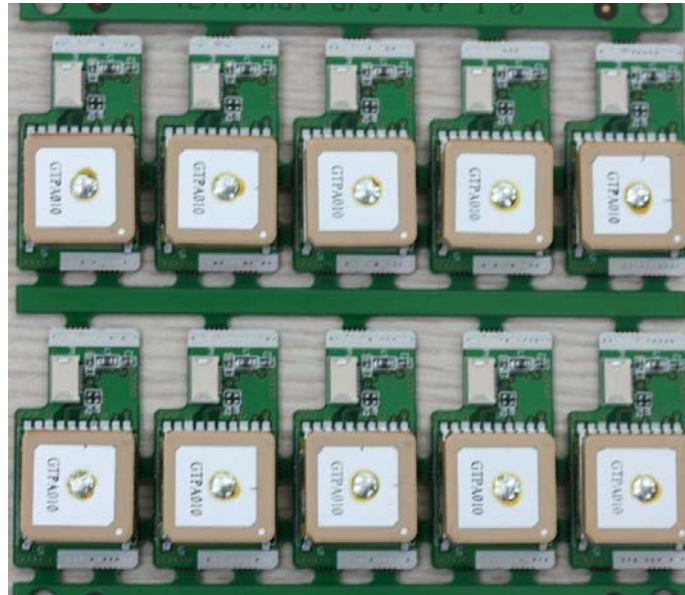


Figure below shows visible antenna shifts of SMT modules occurred after reflow process resulted from immediate transportation of the PCBA.

Visible Component Move



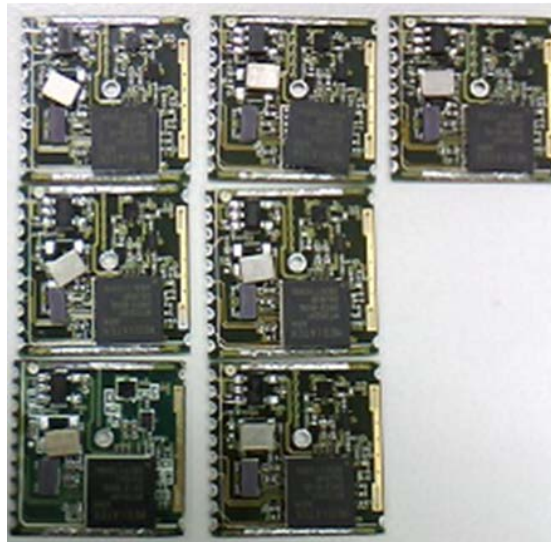
Figures below show module alignment looks perfect from the outside but GPS module failed to function.

After reflow

Looking perfect from the outside



Internal Component Shifts



5. PCBA Handling

Visible parts shifts after module disassembled. Suggestions to PCBA handling after reflow process.

1. After reflow the antenna module keeps heat. Apply cooling by forced air (fan) is recommended until the heat is cooled to the room temperature before handling and transporting.
2. After they are cooled to the room temperature, stack PCBA on anti-static SMT racks to avoid close contacts, as shown in the photo below.



Anti-static racks

6. Module Rework

6.1 Rework on a Standalone Module

Removing solder on the soldering pads of the module with desoldering suction pump tool. Then heat up desoldered area with a heat gun to detach the module.

The following steps describe the best practice for desoldering GlobalTop module from PCBA.

1. Fix the PCBA board on a vice or other fixture. Figure 1.



Figure 1

2. While heating up the soldering pads on the two sides of the module, lift the heat gun above the PCBA about 2 cm apart as shown in Figure 2.



Figure 2

3. While heating the soldering pad around the edge, as the solder slowly gets wet, insert with a blade underneath the module to allow it to lift without too much tilting to avoid shifting of the internal components. When the solder of all solder pads are fully wet, lift up the module gently away from the PCBA surface and allow the module to remain on the blade to allow cooling. Refer to Figure 3.
Caution: Internal component should be hot after heated. Allow the module to remain on the blade approx. 5 seconds to cooling down.



Figure 3

4. Clean up the residual solder. On the PCBA main board, use heat gun to apply heat and soldering iron to clean the soldering pads as shown in Figure 4.

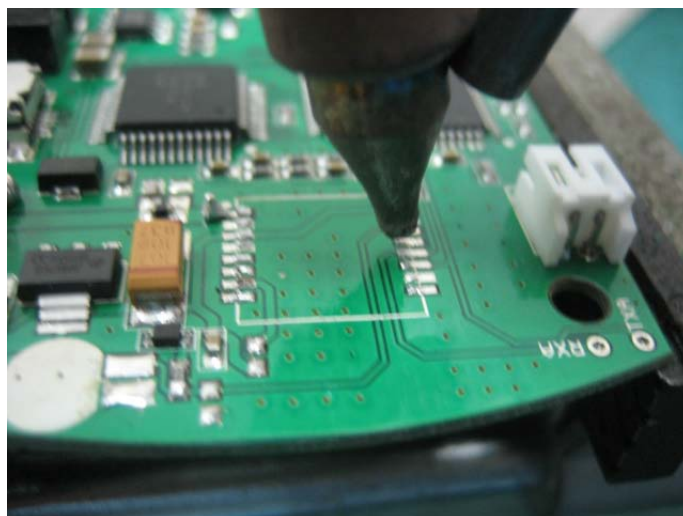


Figure 4

5. Clean residual solder from the pads on the bottom side of the module using heat gun to apply heat and desoldering suction pump tool to clean the pads as shown in Figure 5.



Figure 5

6.2 Rework on a Patch Antenna Module

Applying desoldering braid with soldering iron around soldering pads first to clean the soldering pads. Then remove solder with desoldering suction pump tool while heating up the PCBA with a heat gun around the two edges to detach the GPS module.

The following steps describe the best practice for desoldering GlobalTop antenna module from PCBA.

1. Fix the PCBA board on a vice or other fixture. Figure 1.

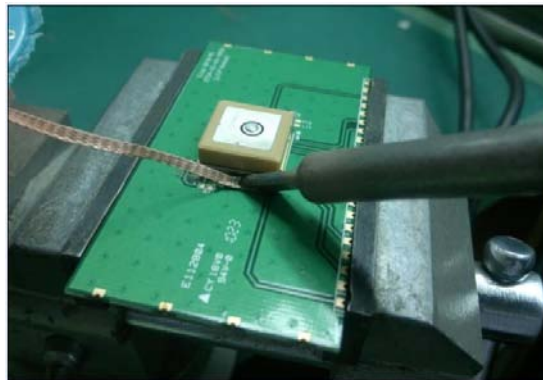


Figure 1

2. Clean solder from the soldering pads using soldering remover like solder wick or desoldering braid as shown in Figure 2.



(圖 2)

3. While heating up the soldering pads on the two sides of the module, lift the heat gun above the PCBA about 2 cm apart as shown in Figure 3.

4. While heating the soldering pad around the edge, as the solder slowly gets wet, insert with a blade underneath the module to allow it to lift without too much tilting to avoid shifting of the internal components. When the solder of all solder pads are fully wet, lift up the module gently away from the PCBA surface and allow the module to remain on the blade to allow cooling. Refer to Figure 4.

Caution: Internal component should be hot after heated. Allow the module to remain on the blade approx. 5 seconds to cooling down.

5. Clean up the residual solder. On the PCBA main board, use heat gun to apply heat and soldering iron to clean the soldering pads as shown in Figure 5.

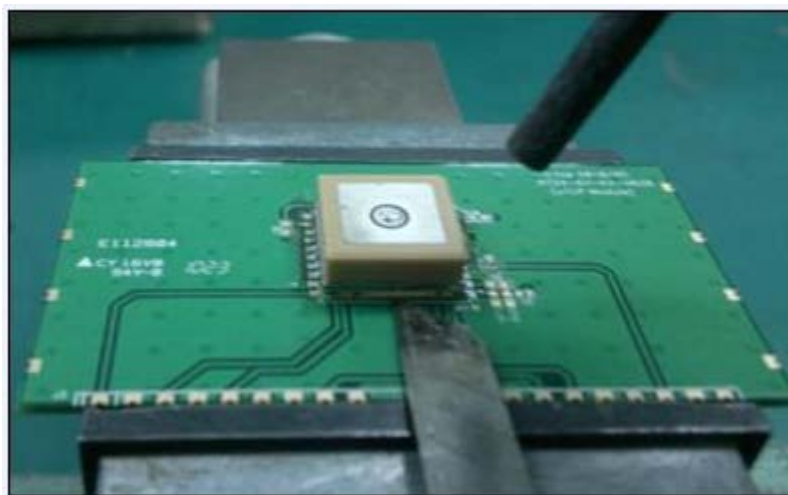


Figure 3

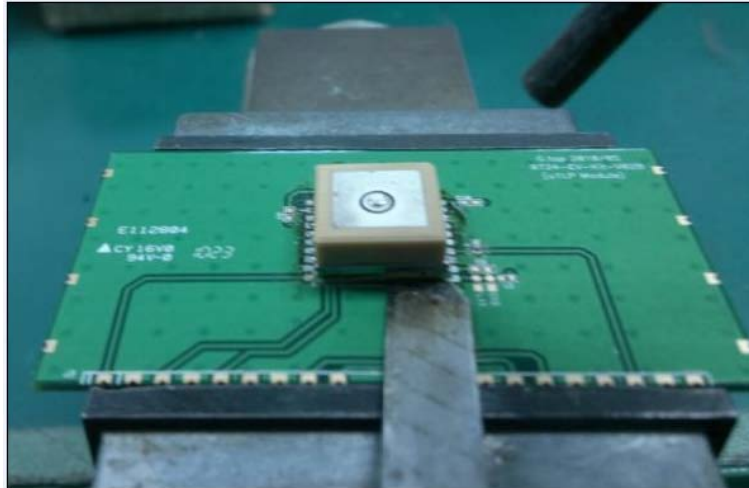


Figure 4

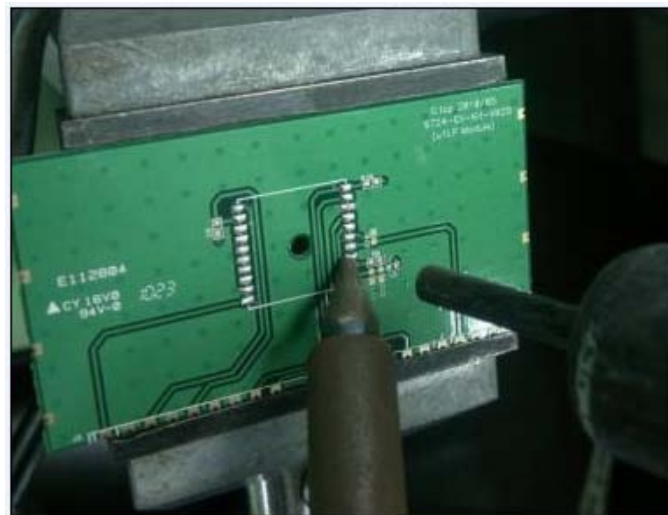


Figure 5

6. Clean residual solder from the pads on the bottom side of the module using solder wick or desoldering braid and soldering iron to clean the pads as shown in Figure 6.

7. Clean residual solder from the antenna hole pad also as shown in Figure 7.



Figure 6

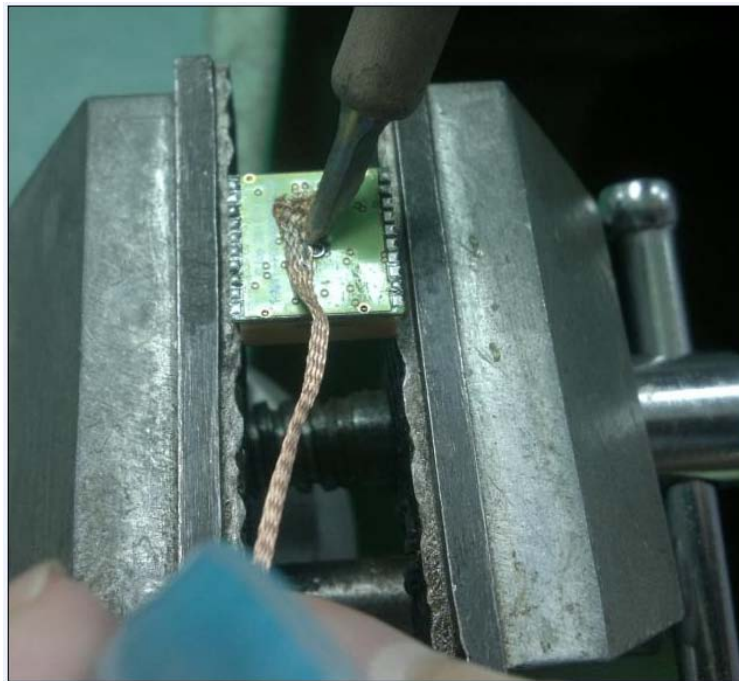


Figure 7

8. Shielding cover and module PCB Separation.

Thoroughly remove and clean solder from along side of the Shielding cover which holds the Shielding cover and the PCB of the module together, refer to Figure 8. Use knife blade to split the attachment as shown in Figure 9. Figure 10 shows the internal component of the module after the Shielding cover is detached.



Figure 8



Figure 9

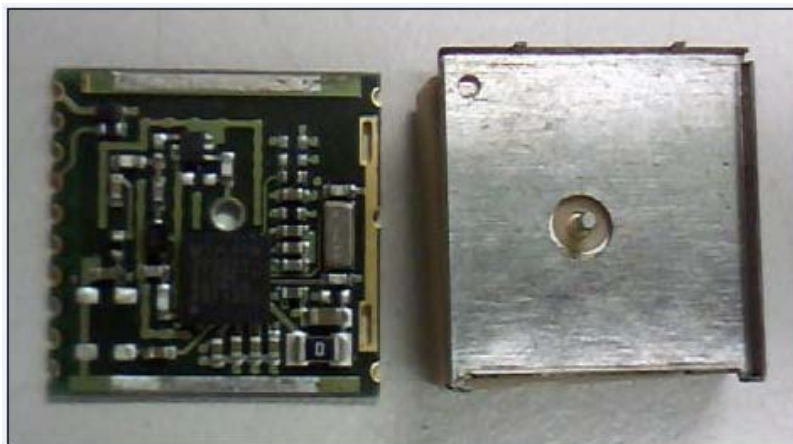


Figure 10