menlomicro

MM5620/MM5622 LGA device

Assembly Guidelines Rev2.2

September 2024

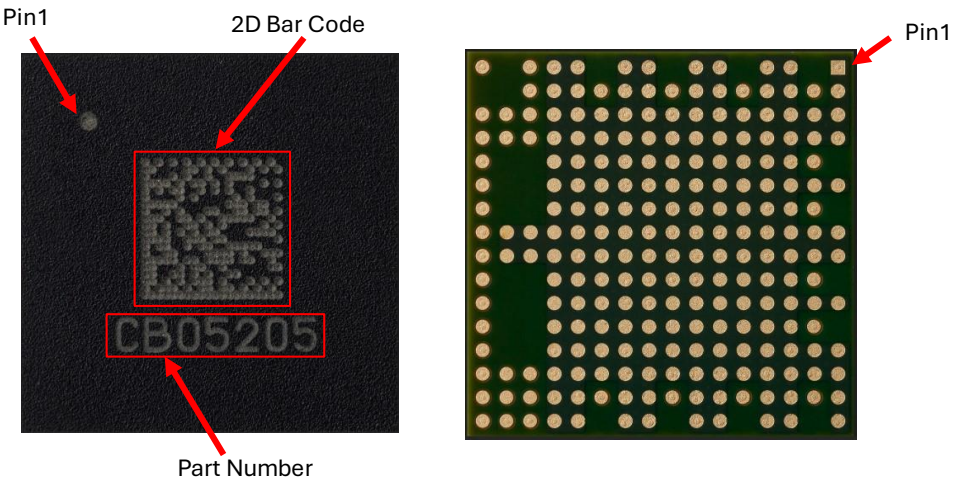
Introduction

This application note provides general guidelines for the assembly of the MM5620/MM5622 land grid array (LGA) package.

MM5620 LGA Device

Figure 1 shows the topside and bottom side of the LGA package, typical part marking including machine-readable 2D bar code and human-readable part number.

The human-readable part number is CBxxxxx for the MM5620 device and CCxxxxx for the MM5622 device.



(a) Top Side *(b) Bottom Side*

Figure 1: Marking with Pin 1 identifier, 2D barcode, Part Number, and Pads

MM5620 Package Outline Drawing

Figure 2 shows the Package Outline Drawing. Pin 1 is a square pad which dimension is 0.3 mm x 0.3 mm

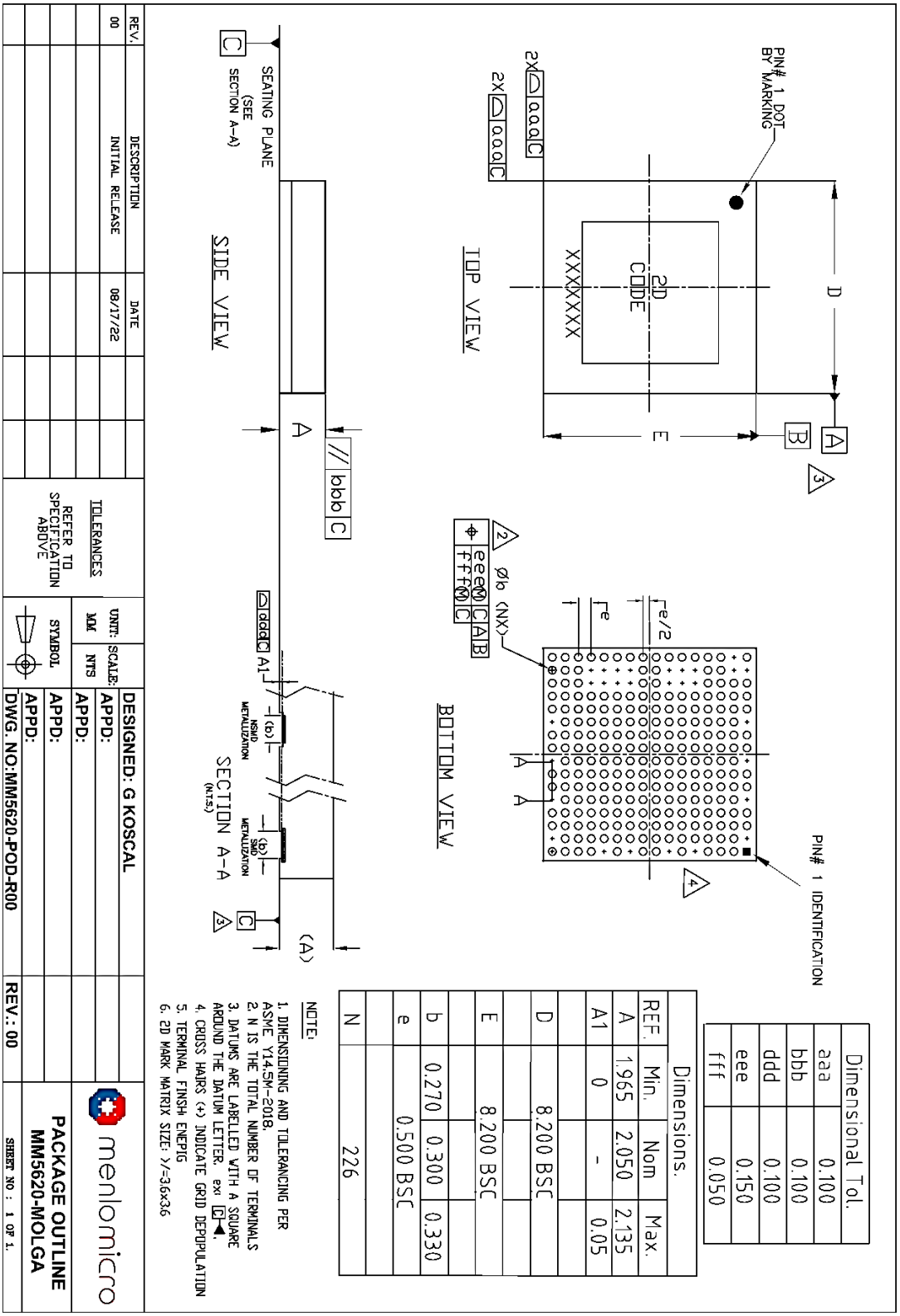


Figure 2: 8.2 x 8.2 mm2 LGA Package Outline Drawing

The devices are provided in tape and reel as defined below in Figure 3.

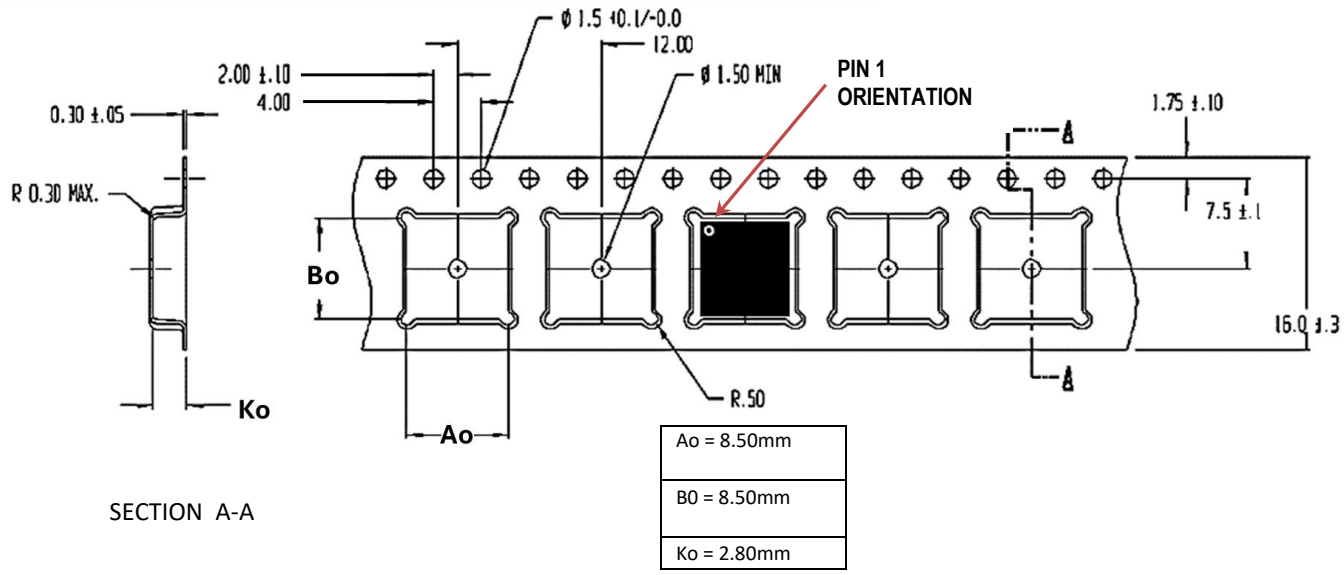


Figure 3: Tape and Reel definition, MM5620/MM5622 (dimensions in mm)

PCB Board design considerations

For the best results, an RF substrate should be chosen based on the operating frequency, operating parameters, and cost considerations for the design.

- The PCB pad diameter is 0.33 mm for SMD and 0.30 mm for NSMD.
- While no particular surface finish is required, Menlo has used both Electroless Nickel Immersion Gold (ENIG) and Electroless Palladium Autocatalytic Gold (EPAG) successfully.
- Use a typical 0.020 mm thick solder mask over circuit traces.
- Use a non-conductive silkscreen border outline around the part location to aid in part placement during assembly and any possible rework.
- Selection of NSMD and SMD should be based on the complexity of a specific board design and the PCB fabrication house's capability for solder mask registration and its tolerance. The SMD and NSMD pads are shown in Figure 4 and Figure 5.

See MM5620/MM5622 EVK design files for reference.

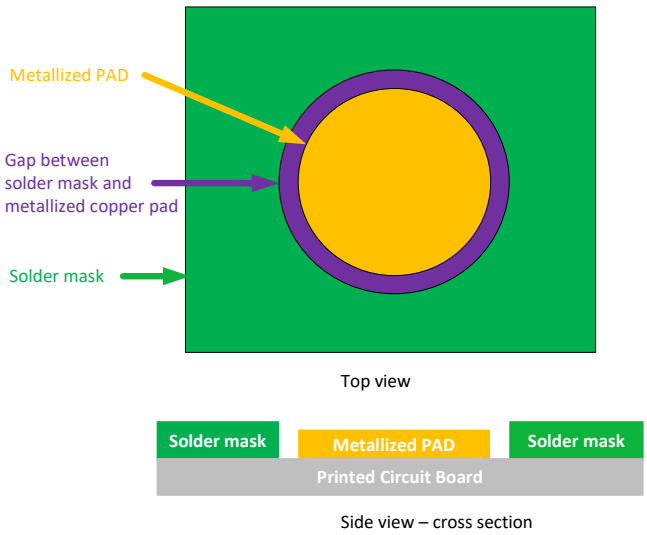


Figure 4: Non-Solder Mask Defined Pad (NSMD)

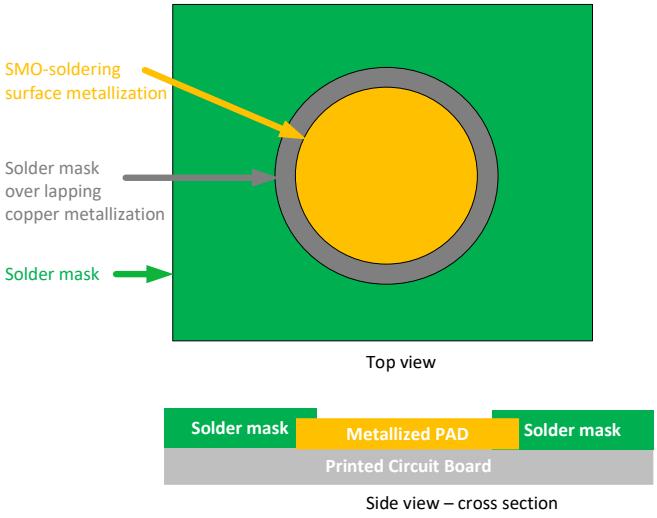


Figure 5: Solder Defined Pad (SMD)

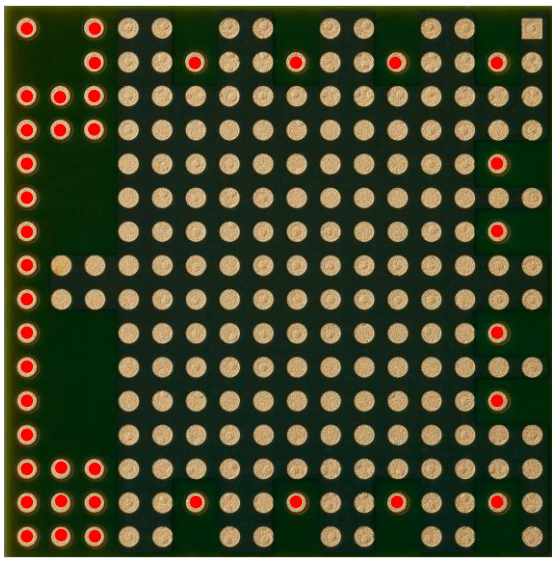


Figure 6: NSMD pads on the package

- Figure 6 shows NSMD pad locations as viewed from the bottom surface of the package. The red dot signifies the location of 39 NMSD pads. All unmarked pads are of SMD construction.

Solder Screen Recommendations

- Each connected pad on the device needs to be soldered to the PCB substrate. The solder stencil Menlo uses is 0.1016 mm (4 mil) thick stainless steel, with openings of 0.2921 mm in diameter (Figure 7). Stencil opening for NSMD pad may need to be larger than that of SMD pad.
- A laser-cut, and electro-polished stencil.
- No nano-coating is required on the stencil.
- Lead-Free No-clean Type 4 solder paste, Indium Corporation 10.8HF SAC305 was used in our trials. Menlo has run small volumes successfully using type 4 solder paste but recommend using type 5 in high volume manufacturing (HVM) for better printing consistency.

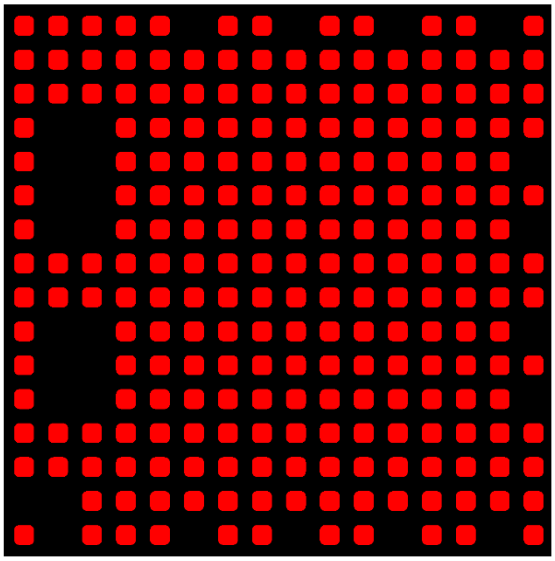


Figure 7: Typical stencil design

Pick and Place Procedure

- Treat the device as electrostatic sensitive and observe all customary handling precautions including working on static dissipative surfaces, wearing wrist/shoe straps, ESD smocks/jackets, or other ESD control devices. Store unused devices in their packaging in ESD bags. Do not store loose parts in bulk bins, do not use parts that have been stored on workbench tops or that have been dropped.
- The MM5620 device is a Moisture sensitive level 3 (MSL3). Any parts removed from the vacuum bag should be assembled on the PC board within no more than 168 hours. Parts exceeding 168 hours should be baked at 125°C ±5°C for 12 hours before assembly.
- To ensure centering of the part during pick and place use “look up” vision centering.
- A vacuum picker tool shall be used to place the part.
- Component placement force should not exceed 100 grams.

Solder Profile

Before board assembly, it is recommended to run the solder profile with thermocouples at the part location to ensure reflow temperature/duration are reached. This is especially important with large dense boards that contain many parts.

While determining the solder profile, it is possible to adjust the solder temperature reflow profile, stencil thickness, aperture opening, etc. based on the specific PCB design, especially for a thick and large board.

Solder Profile for Lead-free Solder

Figure 8 details the recommended solder profile for lead-free solder.

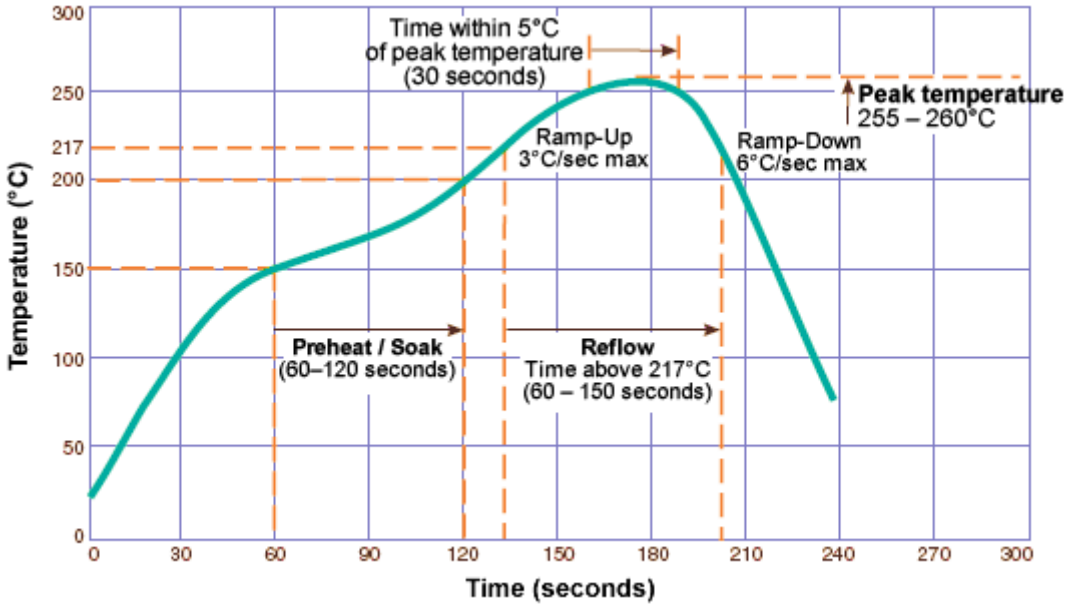


Figure 8: SAC305 solder reflow profile

A ROHS-compliant Solder Alloy used is SAC alloy: 96.5% Sn, 3.0%Ag, 0.5%Cu. These are the nominal percentages of the components. This alloy is designed to replace SnPb solders to eliminate Lead (Pb) from the process, requiring a higher reflow temperature. Moisture resistance performance may be impacted if not using the Pb-Free reflow conditions.

Follow Moisture Sensitivity Level (MSL) 3 handling precautions specified in IPC/JEDEC J-STD-020, latest revision.

Solder Profile for Leaded Solder

Figure 9 details the recommended solder profile for leaded SnPb solders. Before board assembly, it is recommended to run the solder profile with thermocouples at the part location to ensure reflow temperature/duration are reached. This is especially important with large dense boards that contain many parts.

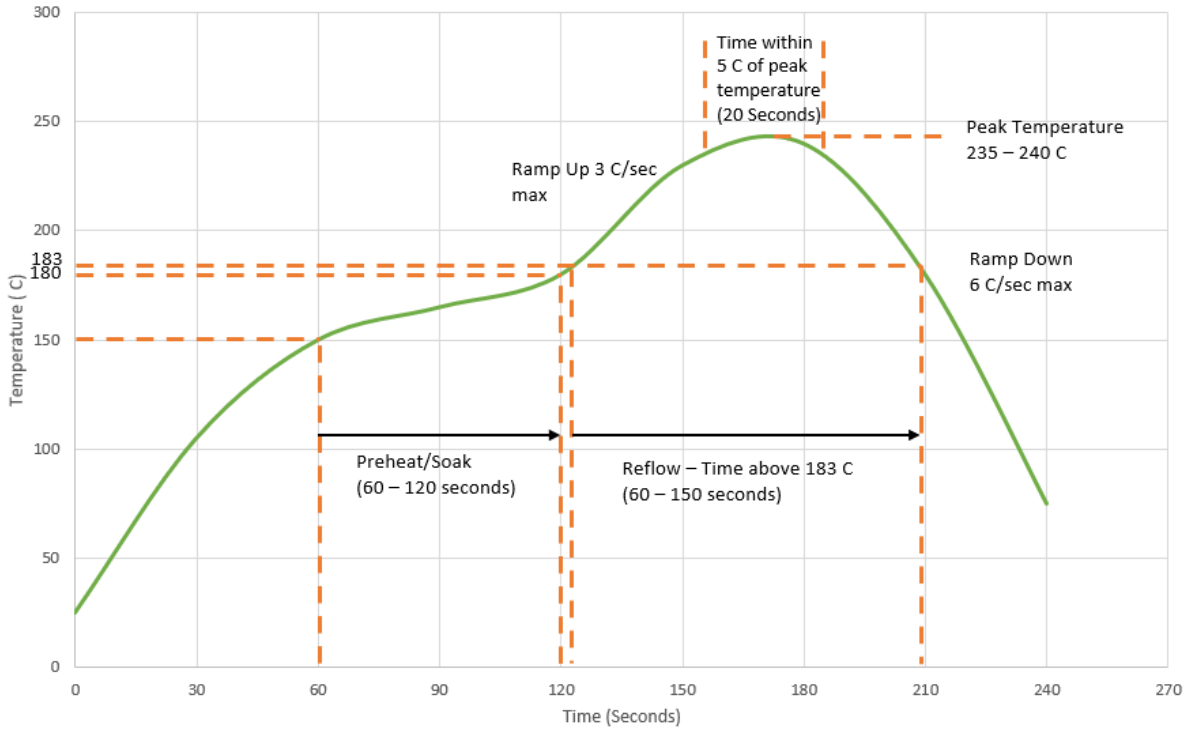


Figure 9: Leaded/Eutectic Solder reflow profile

PCB Cleaning

If no-clean solder paste is used, PCB cleaning is typically not required. Heated deionized water has been used successfully. In addition, Ultrasonic cleaning is not recommended and may cause part damage/reliability issues.

Solder Joint Inspection

Post-flow solder joint inspection should be done using optical inspection to detect any faulty solder joint profile, shorts, misalignment, etc. An X-ray inspection is recommended.

Refer to the IPC-A-610 document which was written for Acceptability of Electronic Assemblies.

Board Assembly Checklist

Below is the board assembly check list with Menlo's recommendations.

	Item	Recommended
1	Material CTE	<17 ppm/C
2	# Layers and total thickness	Note, thermal load of thick boards may require extended soak during reflow. Profiling of board is recommended
3	Component density	Care should be taken with component layout as rework may be difficult
4	Pad opening	330um for SMD/300um for NSMD
5	Pad - SMD/NSMD	SMD
6	Solder paste (Leaded/Unleaded)	SAC305 lead-free solder
7	Solder paste type	5
8	Stencil thickness	4 mil/100um
9	Stencil opening	0.2921 mm in diameter
10	Stencil coating	Electropolish and nanocoating
11	Reflow max temp near device	260 °C
12	Reflow duration above liquidus near device	30 seconds
13	Solder mask thickness	20 um typical
14	Post assembly board test	X-Ray. On resistance can be measured on HS1x-MS1x-LS1x to ensure operation. Similarly, HS2x-MS2x-LS2x
15	ESD Controls - Board Assembly, instrument assembly	Class 0 device.
16	ESD Controls - In production	Class 0 device.
17	Flux type	Clean or No clean
18	Flux cleaning method	None or Hot Aqueous wash