Solder Paste Printing Directly To Land Grid Array (LGA) and Quad Flat No-Leads (QFN) Packages

Unlike Ball Grid Arrays (BGA), Land Grid Array (LGA) and Quad Flat No-Leads (QFN) packages are supplied with bare plated contact pads and solder must be used in the assembly or rework process to install such components. There are several methods of how solder may be applied such as paste printed on the PCB, pre-tinned on the package, or applied directly to the package prior to reflow. The benefit of applying solder paste directly to the package versus pre-tinning is that it reduces processing time, eliminates unnecessary handling, and does not subject a part to an additional thermal cycle. This guide outlines step by step procedure of solder paste being transferred directly to an LGA device prior to being optically aligned and soldered using the AT-GDP SMD/BGA Placement & Rework Station.

Step 1
A laser cut stainless steel stencil is secured in the machine’s holder.
Step 2
LGA is picked up by the machine’s vacuum nozzle.

Step 3
LGA’s contacts are optically aligned over the stencil’s apertures.
Camera view of the stencil apertures.

Camera view of the LGAs contact pads.
Camera view of the stencil apertures when misaligned relative to the LGAs contact pads.

Camera view of the stencil and LGA aligned after fine adjustment of the positioning stage.
Step 4

After being aligned, LGA is lowered down to the stencil via machine’s precise auto Z axis slide with measurable placement force control.

Step 5

A squeegee is used to transfer solder paste directly onto the LGAs contact pads.
Step 6

LGA is moved up vertically to the home position and is now ready for optical alignment / installation to the PCB. Solder paste has been successfully transferred to the contact pads and can be inspected at high magnification prior to initiating installation.