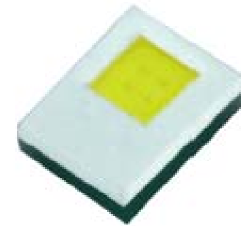
DOMINANT™
Opto Technologies
Innovating Illumination

APPLICATION NOTES:

Surface Mounting And Substrate Selection Guide For NagaJo 2025

Introduction:

NagaJo 2025 is a very compact light source (2.0 x 2.5 x 0.85 mm) pack with high intensity to enhance design flexibility. It is particular suitable for application where space is a major constrain.



- Super high brightness surface mount LED
- 120° viewing angle.
- Compact package outline (LxW) of 2.0 x 2.5 mm.
- Ultra-low height profile - 0.85mm.
- Low thermal resistance, Rthjs:4.3K/W
- Electrical Insulated Thermal Pad to enhance heat dissipation
- Compatible to IR reflow soldering.
- Environmental friendly; RoHS compliance.
- Superior corrosion resistance.
- Compliance to automotive standard; AEC-Q101.

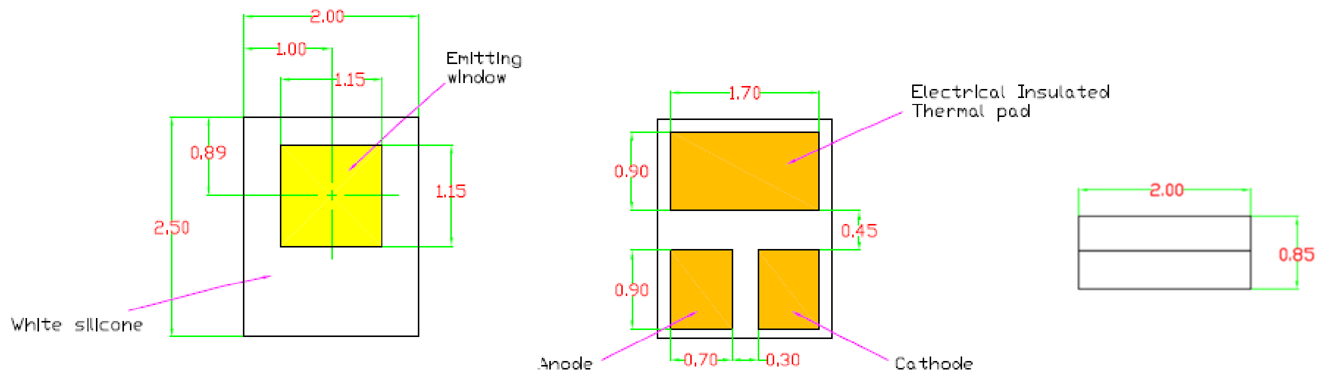


Figure 1: NagaJo 2025 Package Dimension

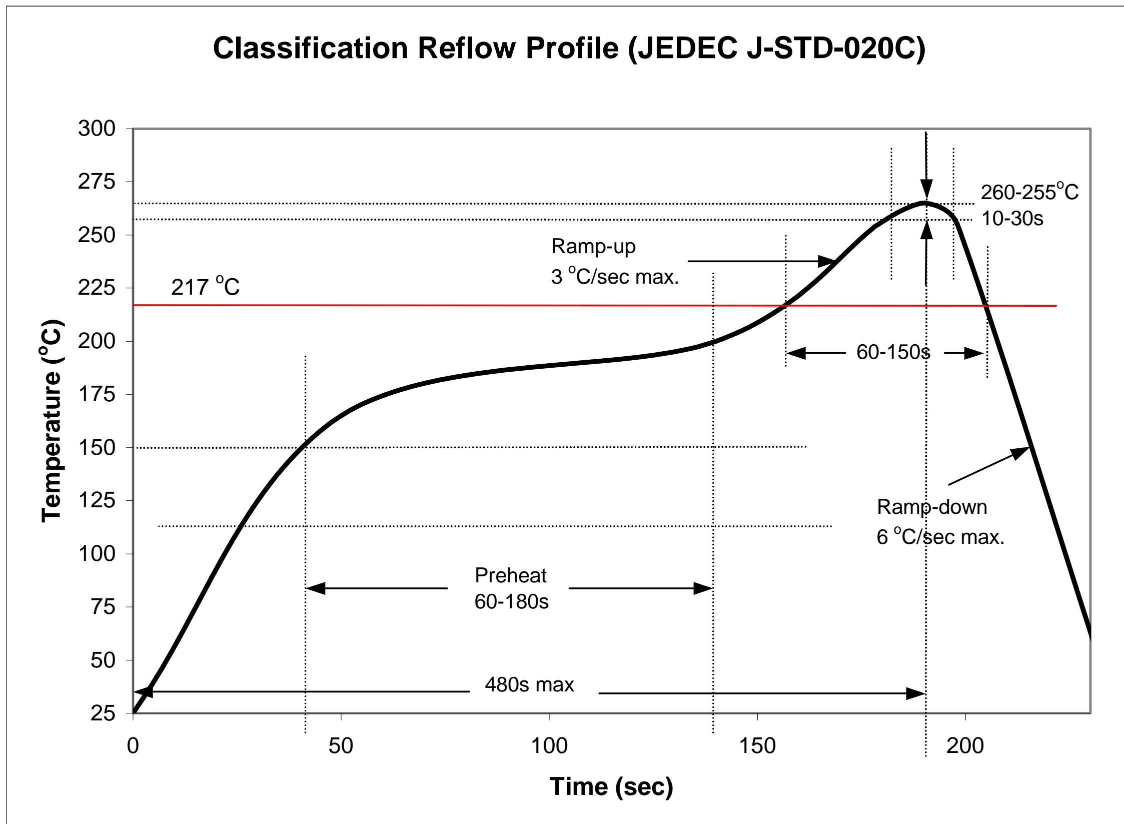
Standard Soldering Process:

The NagaJo package soldering surfaces are plated with gold (Au) and are therefore RoHs compliant. The component is designed to be compatible to the existing industry SMT process and IR-reflow.

However, due to the unique design, all the soldering terminals are located at the bottom surface of the component. This greatly reduces the space required and also enhances the thermal dissipation capability of the component. Heat from the LED chip is directly conducted via the soldering terminals to the external environment. Thermal path is kept to the very minimum.

As for the soldering process, the component is qualified for Pb-free soldering profile. The profiles is as per described in the datasheet.

Recommended Pb Free IR-Reflow Soldering Profile



Surface Mounting – Factors to Consider:

This application note provides a guideline for the surface mounting of NagaJo LED. The following parameters have to be considered in order to optimize the surface mounting performance.

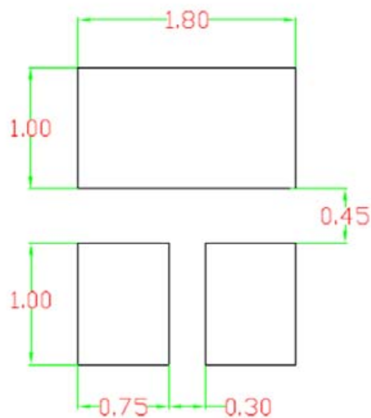
- Solder pad size
- Solder stencil size
- Pipette (nozzle)
- Solder paste thickness

Solder Pad Size

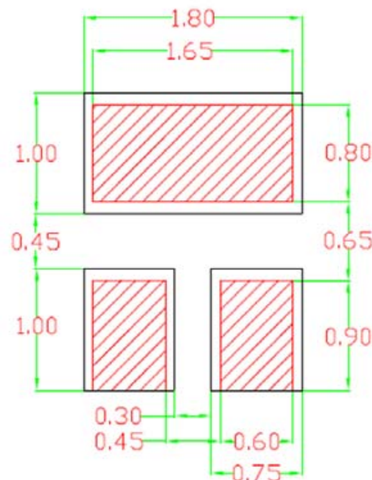
The recommended solder pad design is as illustrated in the data-sheet.

NagaJo solder lead and stencil design:

Recommended Solder Pad



Recommended Solder Stencil Design



Solder Stencil

In order to minimize solder bridging problems, it is common to design stencil aperture size smaller than the recommended solder pad. Excessive amount of solder paste deployed will result to tilted parts and inaccurate placement position. It is recommended that the aperture is reduced to 75% of the recommended solder pad design.

Pipette

Pick and place machine should be able to process NagaJo devices with the required placement accuracy. Care should be observed that the surface of the pipette which is in contact with the LED is flat and smooth. Because NagaJo device construction is different compare to conventional LED package, the pick up position of the pipette should be on the hard phosphor plate as depicted in the photo below. Both vacuum suction force and placement force of the pipette should be <5N/mm^2 during SMT to prevent damage to the LED

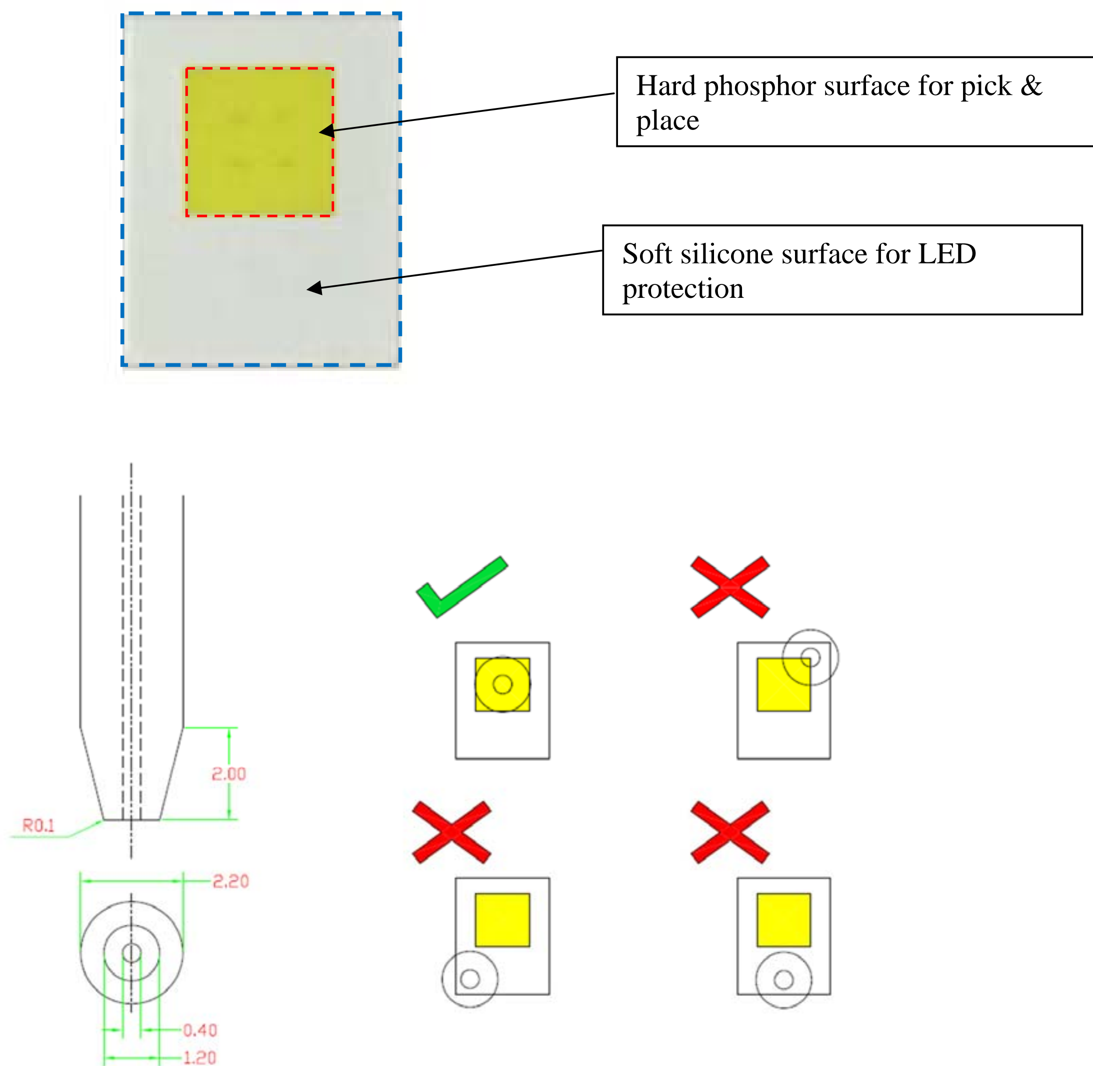


Figure 2: Recommended LED Pick & Place Nozzle Size And Pick Up Position

Solder Paste Thickness

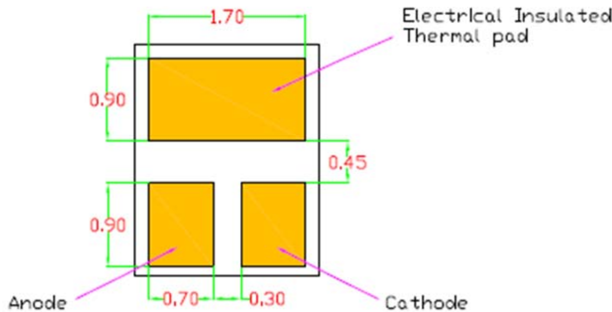
We recommend using minimum solder paste in order to achieve a good solder formation. A solder paste thickness of 0.125 mm will be optimum.

Solder Paste Type

Dominant has tested the Alpha P39 Maxrel and Henkel Innolot (90iSC) base solder paste with satisfactory results. However, since application environments vary widely, we recommend that customers perform their own solder paste evaluation in order to ensure it is suitable for the targeted application.

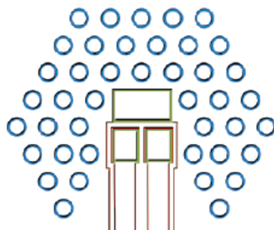
Printed Circuit Design For Enhance Heat Dissipation

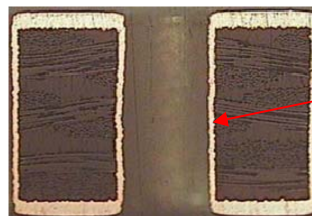
For NagaJo package, the base substrate is built with Aluminum Nitride of 170W/mk thermal conductivity, so all the lead can be primary thermal path to carry heat away from the package. Copper coverage dimension on PCB board can be extended beyond the solder pad in order to improve heat dissipation.



For NagaJo package, the top solder pad is electrically insulated, thus provide more freedom to designer to utilize either method below to enhance package heat dissipation:

(1) Dual Sided FR4 PCB with copper plated/plug through hole

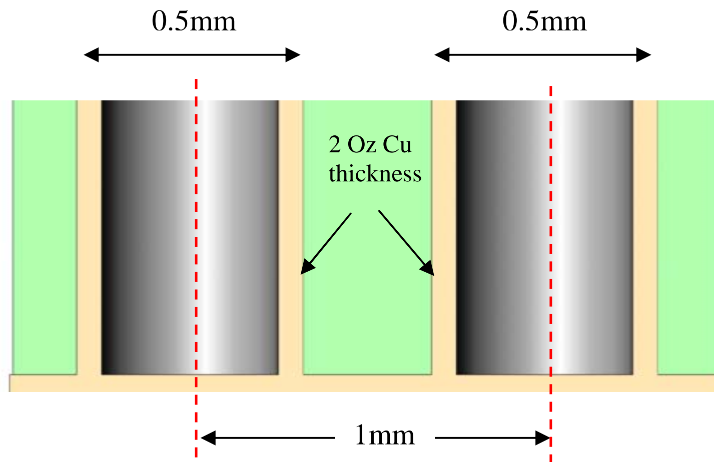




Continuous layer of copper from top to the bottom of PCB for heat conduction

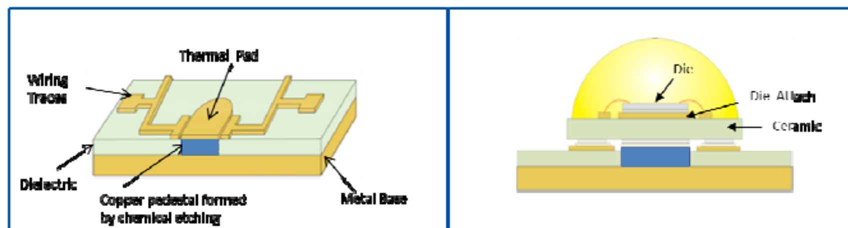
Copper plated through hole via can be added to the inexpensive dual side FR4 PCB. The Electrical Insulated lead on the LED package is soldered to the electrical insulated pad on the PCB. The top copper layer of the PCB is connected to the back of FR4 copper layer through the copper plated via hole. By using this construction, maximum thermal conductivity can be achieved direct from the front to the back of the PCB.

The recommended through hole diameter is 0.5mm and the hole to hole pitch of 1mm. The proposed plated through hole copper thickness is 2Oz (~ 70um). All the recommended specification above is well within typical PCB fabrication capability in order not to incur extra cost or yield loss in standard PCB manufacturing process.



(2) MCPCB With Isolated Thermal Post

With the new development in MCPCB process, now it is possible to selectively laminate FR4 pcb on the etched/stamped aluminum/copper core. The LED electrical isolated thermal pad can be directly soldered to the bulk aluminum/copper core. This new concept eliminated the thermal bottleneck faced by conventional MCPCB which insulator layer is sandwiched in between laminated FR4 and metal core.



Available Metal Base: Copper (390W/m°C) and Cu-Aluminum Alloy (15/85 Cu/Al, 230W/m°C)

Typical Performance	Copper Base	AL Alloy Base
Electrical Thermal Resistance Junction To Solder, Rthjs (K/W)	2.5	2.5
Electrical Thermal Resistance Junction To Case, Rthjc (K/W)	3.6	4.8

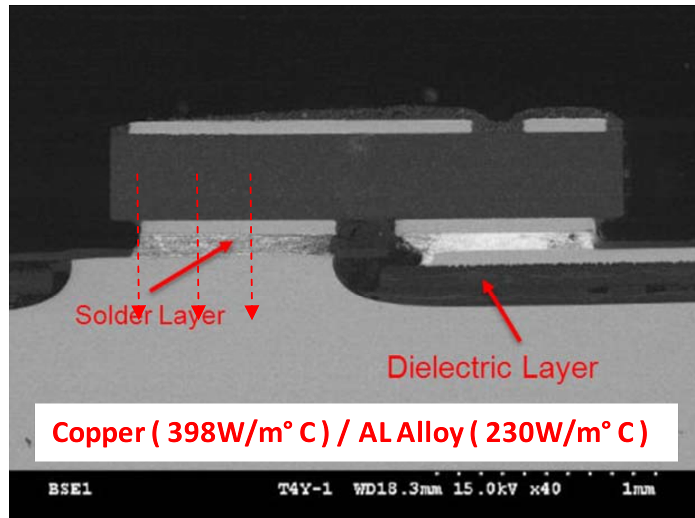


Figure 3: Direct Thermal Path From LED Insulated Thermal Pad To Copper Base Of MCPCB

Solder Quality Check After SMT Process

For NagaJo LED, the primary soldering surfaces are at the bottom of the LED component. Thus no solder fillet will be observed on the part after SMT. Current industry practice is to install AOI after solder paste printing to ensure solder paste is perfectly print on pad before component landing. Alternately, inline X-ray can be used for solder void check after SMT process.

LED	Void Percentage		
Nagajo 2025 (Single Chip) Specification Limit : < 25% (CPK > 1.67)	2.40%	2.03%	2.69%

Figure 4: Inline X-Ray Solder Void Inspection After SMT

Storage Method After SMT

For PCB assembly that mounted with NagaJo 2025, it should not be stack together after IR reflow, else it would have high chance of damaging the NagaJo 2025 LED. Recommended method is have a dedicated carrier so that each PCB assembly is with at least 5mm away from each other

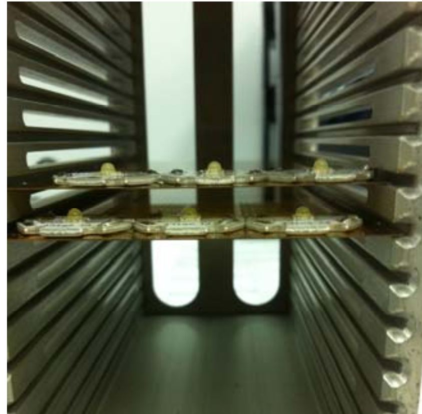
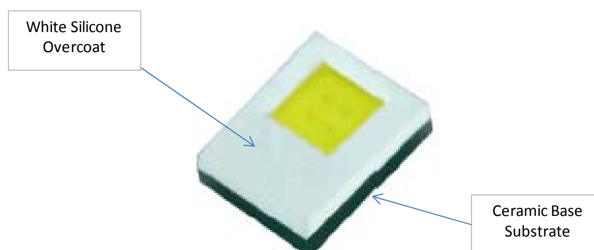


Figure 5: No stacking of LED is allowed after SMT

Handling Precautions

- Mechanical forces exerted onto the white silicone overcoat layer of NagaJo LED should be minimized.



- For manual handling, anti-static/conductive plastic tweezers should be use, always pick up from the sides of the ceramic substrate and never from the sides where white silicone will be pressed. Handling with metal tweezers can possibly inducing package crack at ceramic layer.

- A better alternative for manual handling of NagaJo package is using vacuum pen. The suction tip should be made of a soft material such as rubber to minimize the mechanical force exerted onto the top surface of the white silicone layer. Care should be taken to avoid the soft material from contaminating the top side surface of the LED emitting area.

- In order to avoid any mechanical damage to the LED, do not apply more than 5N of shear force (500g-f) on the LED package

- Do not touch the top surface of LED with fingers or apply any pressure to it when handling finished boards containing NagaJo LED