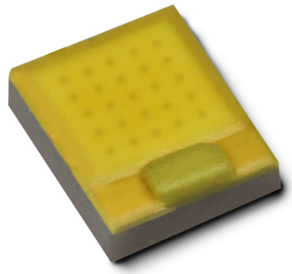
LUMILEDS

GENERAL ILLUMINATION

LUXEON Z ES

Assembly and Handling Information



Introduction

This application brief addresses the recommended assembly and handling procedures for LUXEON Z ES emitters. Proper assembly, handling, and thermal management, as outlined in this application brief, ensure high optical output and long lumen maintenance for LUXEON Z ES emitters.

Scope

The assembly and handling guidelines in this application brief apply to all the products in the LUXEON Z ES product family. In the remainder of this document the term LUXEON ES emitter refers to this product family and the term LUXEON emitters refers to a generic Lumileds LED.

Table of Contents

1. Component

1.1 Description

The LUXEON Z ES emitter is an ultra-compact, surface mount, high-power LED. Each LUXEON Z ES emitter consists of a high brightness InGaN chip with a phosphor layer on top of a ceramic substrate. The ceramic substrate provides mechanical support and provides a thermal path from the LED chip to the bottom of the emitter. An interconnect layer electrically connects the LED chip to cathode and anode pads of equal size on the bottom of the ceramic substrate. The thermal pad, anode and cathode of the LUXEON Z ES emitter are shown in Figure 1.

The entire top surface of the LUXEON Z ES is covered with a thin layer of silicone to shield the chip from the environment and to create white light.

LUXEON Z ES emitters contain a transient voltage suppressor (TVS) chip which protects the LED chip against electrostatic discharge (ESD) events. The TVS chip creates some minor topographical variations across the top surface of the LUXEON Z ES emitters.

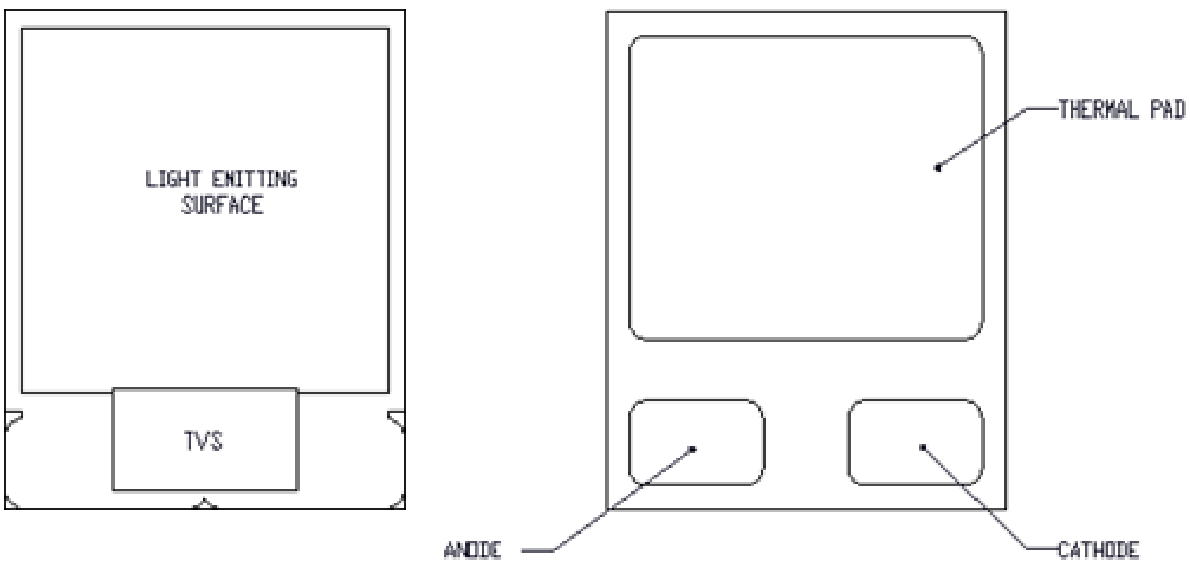


Figure 1. Top view (left) and bottom view (right) of a LUXEON Z ES emitter.

1.2 Optical Center

The theoretical optical center of the LUXEON Z ES emitter is 0.82mm from the top and 0.82mm from the side edges of the ceramic substrate (see Figure 2).

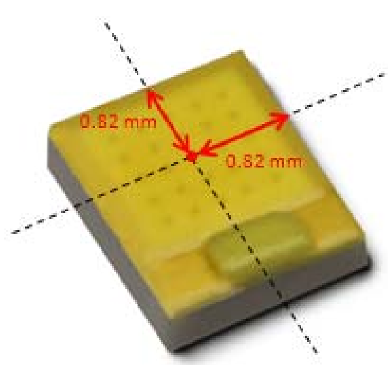


Figure 2. The optical center of the LUXEON Z ES emitter is 0.82mm from the top and 0.82mm from the side edges.

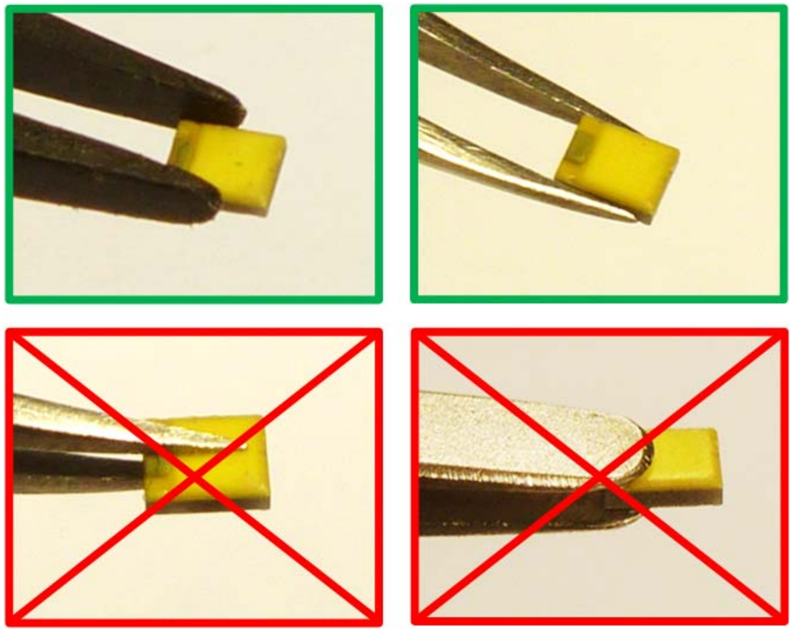


Figure 3. Correct handling (top) and incorrect handling (bottom) of LUXEON emitters.

1.3 Handling Precautions

The LUXEON Z ES emitter is designed to maximize light output and reliability. However, improper handling of the emitter may damage the LED chip and affect the overall performance and reliability. In order to minimize the risk of damage to the LED chip during handling, as will all LUXEON emitters, LUXEON Z ES should only be picked up manually from the side of the ceramic substrate as illustrated in Figure 3.

When handling finished boards containing LUXEON Z ES emitters, do not touch the top surface with any fingers (see Figure 4a) or apply any pressure to it. Also, do no turn over the board for probing, if the electrodes are at the back of the board, or stack multiple boards on top of each other (see Figure 4b). A rough or contaminated surface, being placed against the top of a LUXEON Z ES emitter, may damage the silicone overcoat of the emitter. Furthermore, any pressure applied onto the LUXEON Z ES emitter during probing may damage the silicone layer or the chip underneath.

1.4 Cleaning

The LUXEON Z ES emitter should not be exposed to dust and debris. Excessive dust and debris may cause a drastic decrease in optical output. In the event that the surface of a LUXEON Z ES emitter requires cleaning, a compressed gas duster at a distance of 6" away will be sufficient to remove the dust and debris or an air gun with 20 psi (at nozzle) from a distance of 6". Make sure the parts are secured first.

1.5 Electrical Isolation

The ceramic substrate of LUXEON Z ES emitters electrically isolates the thermal pad from the anode and cathode pads. As a reference the minimum distance between the thermal pad and the anode or cathode is 0.25mm. In order to avoid any electrical shocks and/or damage to the LUXEON Z ES emitter, each design needs to comply with the appropriate standards of safety and isolation distances, known as clearance and creepage distances, respectively (e.g. IEC60950, clause 2.10.4).

1.6 Mechanical Files

Mechanical drawings for LUXEON Z ES (2D and 3D) are available from at lumileds.com.

1.7 Soldering

LUXEON Z ES emitters are designed to be soldered onto a Printed Circuit Board (PCB). For detailed design of PCB, see Section 2.

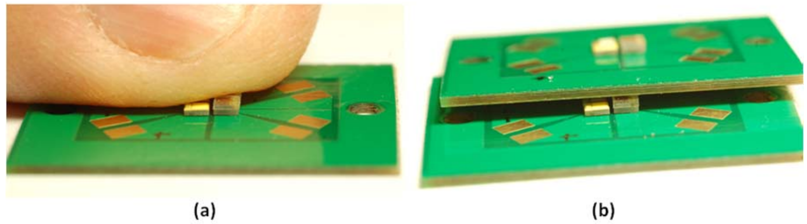


Figure 4. Do not touch the top of surface of the LUXEON emitter when handling a finished board (a) or stack boards with one or more LUXEON emitters on top of each other (b).

2. LUXEON Z ES Printed Circuit Board Design Rules

The LUXEON Z ES emitter is designed to be soldered onto a PCB via standard surface mount technology (SMT). To ensure optimal operation of the LUXEON Z ES emitter, the PCB should be designed to minimize the overall thermal resistance between the LED package and the heat sink.

There are two preferred PCB footprint designs for LUXEON Z ES, depending on the packing density and PCB manufacturing capabilities:

a. Layout design A: for low component count density. The solder mask opening defines the outline of LUXEON Z ES emitter package outline. The PCB pads for LUXEON Z ES emitter are copper trace defined. See Figure 5 (top row).

b. Layout design B: for high component count density or any LED arrangement where it is no longer possible to electrically isolate the thermal pad. Each PCB pad of corresponding LUXEON Z ES emitter pads is solder mask defined. See Figure 5 (bottom row). The higher component count density board may require tighter PCB fabrication tolerance and tighter assembly tolerance such as placement accuracy and stencil printing to ensure high product quality and process yield. Note: layout design B can be also be used for low density assembly.

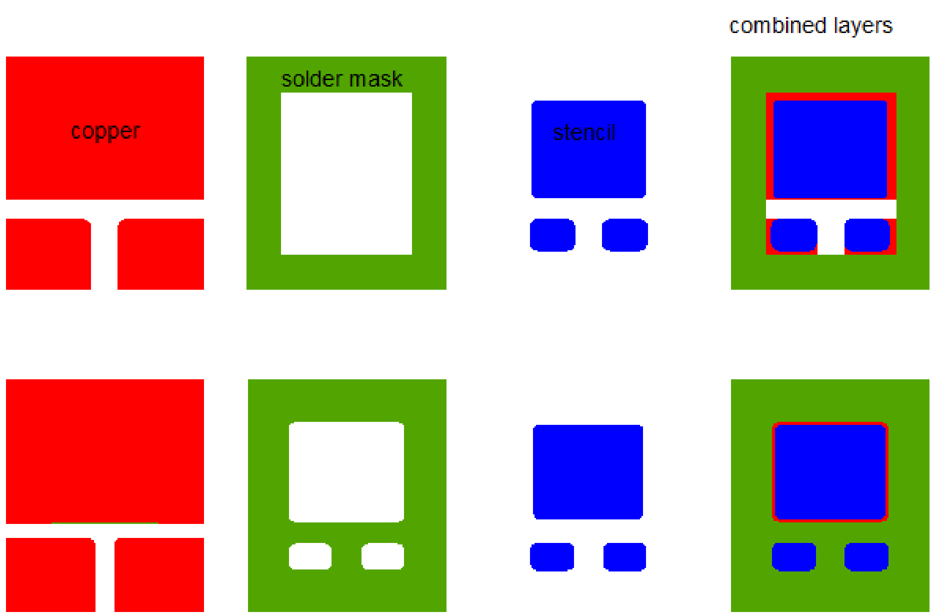


Figure 5. Layout design A (top row) and layout design B (bottom row).

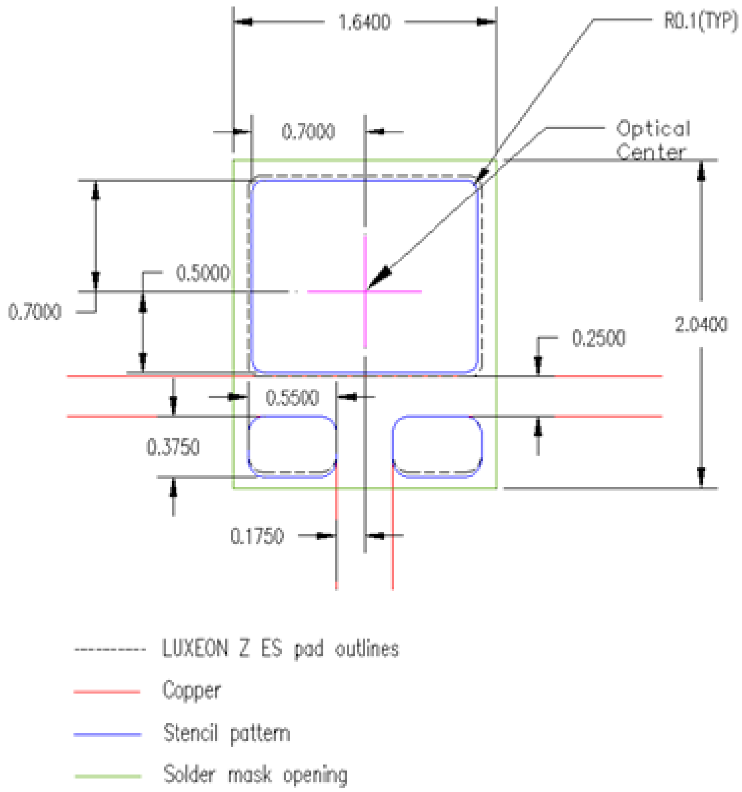


Figure 6. Recommended PCB footprint design for LUXEON Z ES for low component count density (layout design A). Dimensions in mm.

2.1 LUXEON Z ES Footprint and Land Pattern

The LUXEON Z ES emitter has three pads that need to be soldered onto corresponding pads on a PCB to ensure proper thermal and electrical operation. Figure 6 shows the recommended footprint design for a single LUXEON Z ES emitter on MCPCB or FR4 for low density assembly (design A). Heat spreading into the PCB is improved by extending the thermal pad and electrodes on the PCB beyond the package outline which is the same as the solder mask opening as shown in Figure 6. Thermal simulations indicate that heat spreading is maximized if the thermal pad and electrodes are extended 3mm from the center of the LUXEON Z ES emitter.

Figure 7 shows the recommended footprint design for a single LUXEON Z ES emitter for MCPCB for layout design B with solder mask defined pads. In order to efficiently layout the electrical path on a PCB without the use of multi-layer PCB, the thermal pad of LUXEON Z ES emitter can be made electrically active as shown in Figure 8. The solder mask defined pads are required to prevent solder wicking. This is more desirable than a multi-layer PCB since adding layers in the PCB increases the overall thermal resistance of the PCB.

For both design A and B, due to the smaller electrode pad features, the alignment between the stencil and the PCB board during solder paste screen is important to minimize potential shorting or insufficient solder paste between pad and PCB after reflow. For more detail, see section 5.3.

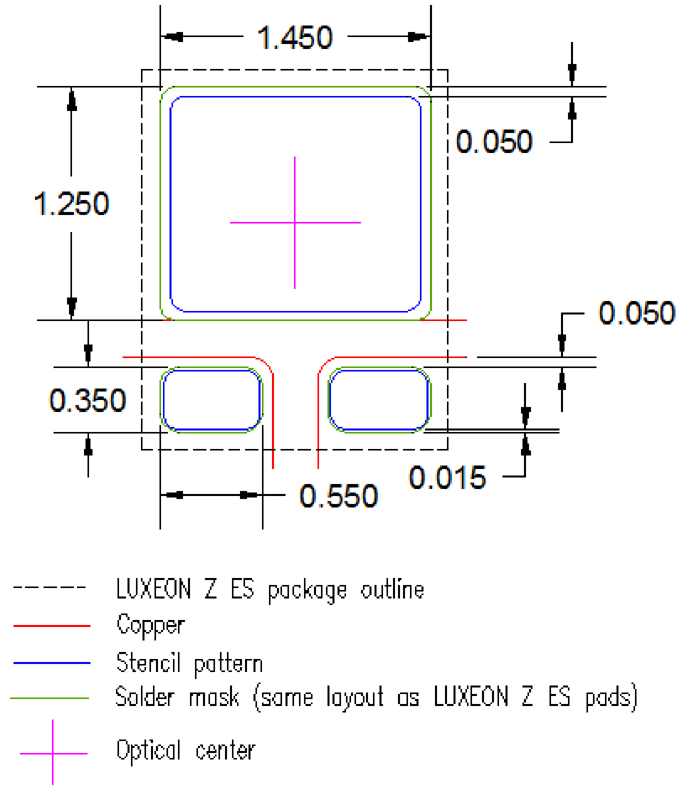


Figure 7. Layout design B. Recommended PCB footprint for LUXEON Z ES for high component count density based on solder mask defined pads. Dimensions in mm.

2.2 Surface Finishing

Lumileds recommends using a high temperature organic solderability preservative (OSP) on the copper layer at this time.

2.3 Minimum Spacing

Lumileds recommends a minimum edge to edge spacing between LUXEON Z ES emitters of 0.2mm for the high density assembly footprint and 0.3mm for the low density assembly footprint. Placing multiple LUXEON Z ES emitters too close to each other may adversely impact the ability of the PCB to dissipate the heat from the emitters. Also, the light output for each LED may drop due to optical absorption by adjacent LED packages.

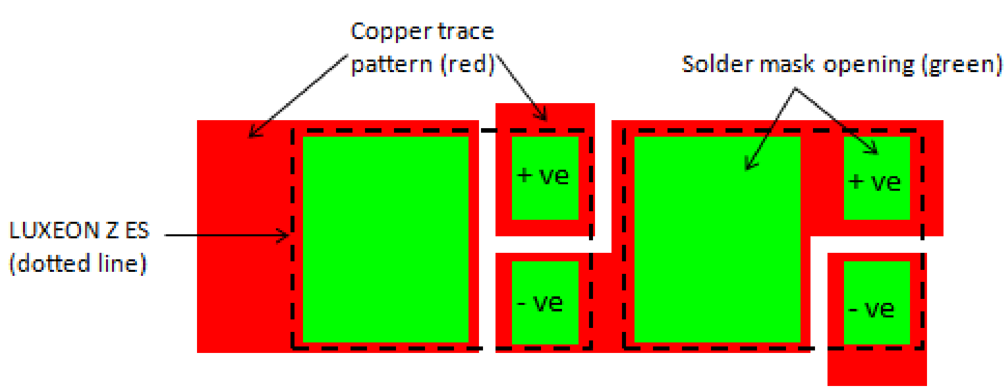


Figure 8. High component count density with layout design B electrically connecting the anode of LUXEON Z ES (right) to the cathode of the adjacent LUXEON Z ES (left) via one of the LUXEON Z ES thermal pads.

3. Thermal Design and Management

3.1 MCPCB Design Parameters

The overall thermal resistance of MCPCB depends on several key factors:

a. Copper layer thickness (in oz such as 1oz versus 2oz copper, thicker copper is preferred)

b. Epoxy dielectric thermal conductivity (in $Wm^{-1}K^{-1}$, a higher value is better)

c. Epoxy dielectric thickness (in um, a lower value is, generally speaking better; note though that the epoxy thickness will impact the PCB dielectric breakdown voltage. If needed, the dielectric breakdown voltage must comply to UL, IEC or any applicable standards in each region or national standards)

d. Aluminum board thickness (in mm, a thinner board is better but may affect the mechanical strength)

e. LED emitter spacing. Smaller spacing increases thermal crowding, resulting in higher thermal resistance values.

A typical cross section of an MCPCB construction is shown in Figure 9.

Throughout this application brief, Lumileds has evaluated a single emitter MCPCB using layout design B with these MCPCB parameters: copper layer of 1oz and 2oz, dielectric thermal conductivity of 2.1 $Wm^{-1}K^{-1}$, dielectric thickness of 100um (yielding approx DC hi-pot test of 4kV) and an aluminum board thickness of 1.0mm.

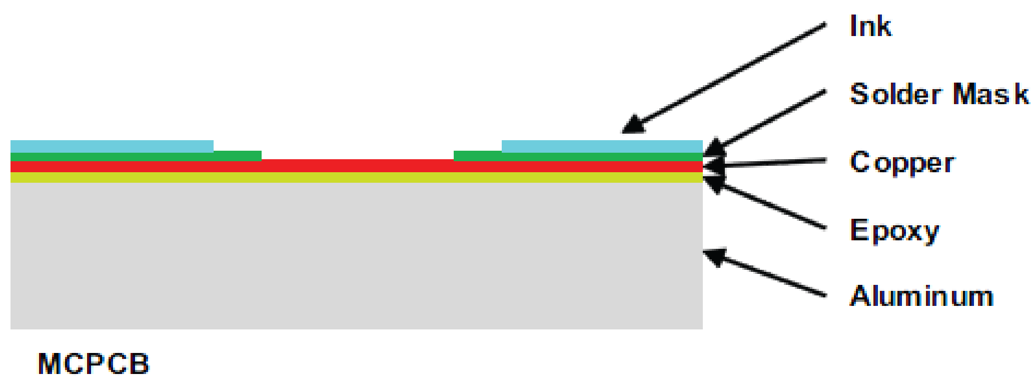


Figure 9. Typical cross section of MCPCB.

3.2 FR4 Design Parameters

The overall thermal resistance of FR4 depends on several key factors:

a. Copper layer thickness plating (in oz, thicker copper is preferred)

b. Via type design (open plated through hole or filled and capped. The latter provides better thermal performance but at higher cost)

c. Number of vias. In general, there is an optimum number of vias and beyond this number, the thermal performance does not increase significantly.

d. Via diameter and spacing. The first row of vias surrounding the thermal pad removes most of the heat so proper positioning and sizing of these vias plays a big factor. The determination of this can be done via thermal simulation.

e. FR4 board thickness (in mm, thinner is better but will reduce mechanical strength)

f. LED emitter spacing. Smaller spacing increases thermal crowding effect, leading to higher thermal resistance.

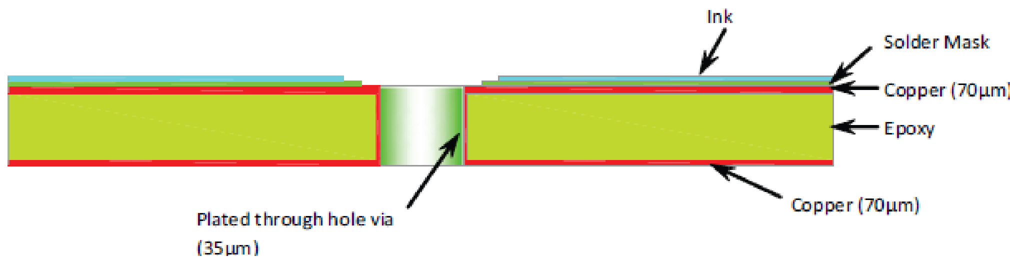


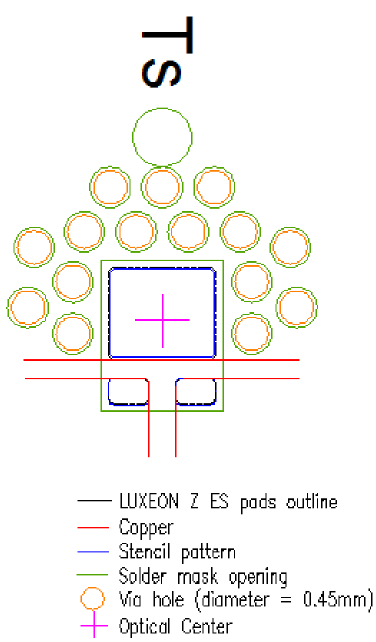
Figure 10. Typical cross section of an FR4 PCB based on an open PTH design.

A typical cross section of an FR4 PCB based on open via plated through hole (PTH) is shown in Figure 10. The LUXEON Z ES footprint on FR4 PCB is similar to design A (see Figure 6) with the exception of thermal vias placed outside the electrically isolated thermal pad. Figure 11 shows an example of a FR4 with PTH which is suitable for low density assembly. There is no recommendation of high density assembly layout design for FR4.

It is not desirable to place open PTH vias on the traces outside the anode and cathode (copper traces must be separated to prevent shorting) since this will require a reliable thermal interface material (non electrically conductive) between the PCB and the heatsink. The quality of this thermal interface material may have impact to the hi-pot dielectric strength test of the PCB to heatsink.

Lumileds has evaluated the thermal performance of the board shown in Figure 11 based on 1.0mm thick FR4 with open PTH vias. The total thickness of the copper plating on the top and bottom of the PCB is 2oz each and the plating inside the thermal vias is about 1oz. The diameter of the thermal vias is 0.45mm (as drilled). This design contains 15 thermal vias.

For general guidelines on FR4 PCB based designs, please refer to section 3 of Lumileds document AB32 "LUXEON LED Assembly and Handling Information".



(X, Y) RELATIVE TO OPTICAL CENTER (MM)		
HOLE NO.	X	Y
1	1.205	-0.175
2	1.205	0.525
3	1.050	1.205
4	0.350	1.205
5	-0.350	1.205
6	-1.050	1.205
7	-1.205	0.525
8	-1.205	-0.175
9	1.811	0.175
10	1.725	0.993
11	0.700	1.811
12	0.000	1.811
13	-0.700	1.811
14	-1.725	0.993
15	-1.811	0.175

Figure 11. LUXEON Z ES FR4 PCB design with 15 open plated-through-hole vias.

3.3 Summary of PCB Thermal Resistance Result

Table 1 shows the typical thermal resistance of various PCB star boards that were evaluated for reference.

Table 1. Typical thermal resistance of PCB star boards (single emitter) between thermal pad and bottom of board/heatsink.

BOARD TYPE (SEE SECTION 3.1 AND 3.2 FOR DETAIL BOARD PARAMETERS)	TYPICAL RTH_PAD-HEATSINK (K/W)
1mm FR4 open PTH vias (1oz + 1oz for additional plating)	9.0
1mm MCPCB 1oz copper (design B)	9.0
1mm MCPCB 2oz copper (design B)	5.6

The MCPCB thermal resistance for design A footprint design is similar to design B.

3.4 High Component Layout Example

Due to its compact and high flux performance in a small area, LUXEON Z ES is well suited for high density packaging applications where multiple LUXEON Z ES emitters are packed right next to each other. There are many different configurations on how the LUXEON Z ES can be placed onto a PCB to meet customer specific application needs. For reference, Lumileds performed thermal studies on two configurations: a 4-up and 9-up configurations as shown in Figure 12 with 200um package to package spacing. The LUXEON Z ES footprint is based on layout design B.

Table 2. Typical thermal resistance from junction-to-heat sink (Rthj-hs) of a 4-up and 9-up Al-MCPCB star boards mounted on infinite heat sink. Include 1-up result for reference. MCPCB is constructed using design B.

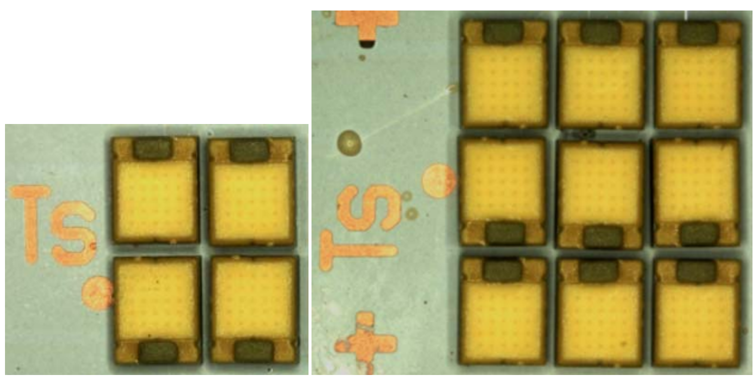


Figure 12. An example of 4-up and 9-up configuration for thermal studies.

EMITTER ARRAY CONFIGURATION (SEE SECTION 3.1 AND 3.2 FOR DETAIL BOARD PARAMETERS USED)	AVERAGE RTH_{J-HS} (K/W) MEASUREMENT	AVERAGE THEORETICAL RTH_{J-HS} (IDEAL)
1-up MCPCB 1oz copper	12.0	12.0
1-up MCPCB 2oz copper	8.6	8.6
4-up MCPCB 1oz copper	5.1	3.0
4-up MCPCB 2oz copper	3.7	2.1
9-up MCPCB 1oz copper	2.9	1.3
9-up MCPCB 2oz copper	2.3	0.95

Table 2 shows the average thermal resistance between junction and heat sink (Rth_{j-hs}) of various Al-MCPCB with 1oz and 2oz copper with various configurations of 1, 4 (2 x 2) and 9 (3 x 3) LUXEON Z ES emitters. The theoretical Rth_{j-hs} of a 4-up and 9-up configurations; assuming each LED is infinitely placed far away, is provided in Table 2 for comparison. This theoretical value limits the lowest possible Rth_{j-hs} that can be achieved when building a 4-up and 9-up board. In high density assembly, thermal crowding effect from adjacent LEDs will affect the overall Rth_{j-hs}, so larger LED spacing reduces this value.

The theoretical formula for this ideal situation can be calculated using the sum of parallel resistance formula i.e.:

$$\text{Rth}_{\text{total}} = 1/(\frac{1}{Rth_1} + \frac{1}{Rth_2} + \frac{1}{Rth_3} \ldots)$$

where Rth_1, Rth_2, etc is the thermal resistance of a single emitter.

Figure 13 shows the graphical result of Table 2 of the Rth_{j-hs} versus the number of emitters on the board with package to package spacing of 200um. An approximate exponential fit curve is shown to illustrate this trend.

The data also show that as more emitters are added, both the 1oz and 2oz copper Rth_{j-hs} converges. Hence increasing the copper layer thickness from 1oz and 2oz for high density configuration (greater than 9 emitters in this example) will not yield significant Rth_{j-hs} improvement.

To further reduce the thermal resistance of such a board, one can consider the following options:

a. Use dielectric material with higher thermal conductivity value
b. Reduce the dielectric material thickness. However this will impact PCB dielectric breakdown voltage
c. Create a direct metal thermal path to the LUXEON Z ES emitter without the use of any dielectric. This is feasible because the LUXEON Z ES thermal pad is electrically isolated from the electrical pads but will result in higher PCB cost
d. Substitute the MCPCB with a highly conductive ceramic material such as aluminum nitride.

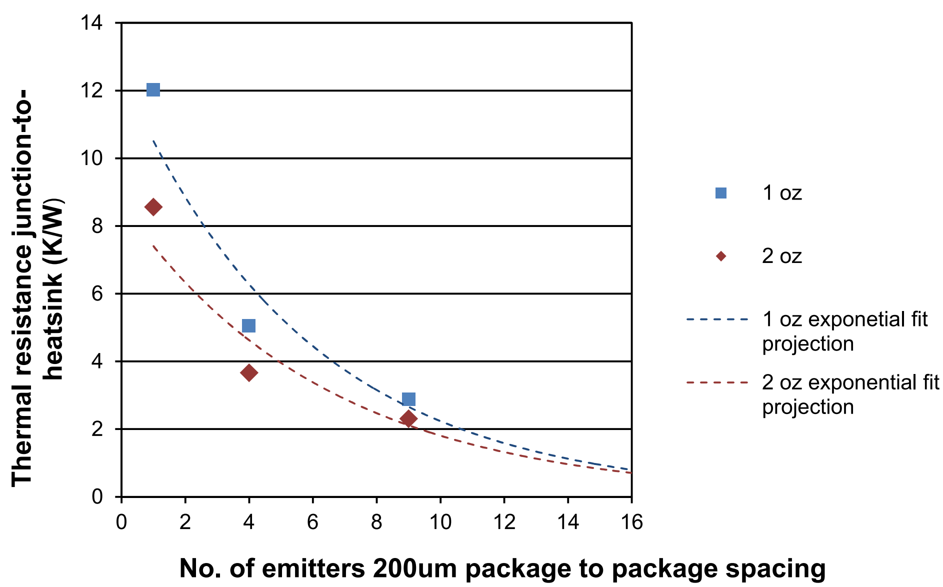


Figure 13. Thermal resistance versus number of emitters.

4. Thermal Measurement Guidelines

This section provides general guidelines on how to determine the junction temperature of a LUXEON Z ES emitter in a stand-alone in order to verify that the junction temperature in the actual application during regular operation does not exceed the maximum allowable temperature specified in the datasheet.

The typical thermal resistance $R\theta_{j\text{-thermal pad}}$ between the junction and the thermal pad for LUXEON Z ES is specified in the LUXEON Z ES datasheet. In LUXEON Z ES, most of the heat flows out from the thermal pad. With this information, the junction temperature T_j can be determined according to the following equation:

$$T_j = T_{thermal\ pad} + R\theta_{j\text{-thermal pad}} \cdot P_{electrical}$$

In this equation $P_{electrical}$ is the electrical power going into the LUXEON Z ES emitter and $T_{thermal\ pad}$ is the temperature at the bottom of one of the LUXEON Z ES thermal pad.

In typical applications it may be difficult, though, to measure the thermal pad temperature $T_{thermal\ pad}$ directly. Therefore, a practical way to determine the LUXEON Z ES junction temperature is by measuring the temperature T_s of a predetermined sensor pad on the PCB right next to the LUXEON Z ES thermal pad with a thermocouple. To ensure accurate readings, the thermocouple must make direct contact with the copper of the PCB onto which the LUXEON Z ES is soldered, i.e. any solder mask or other masking layer must be first removed before mounting the thermocouple onto the PCB. Figure 14 shows the location of a T_s point for various LUXEON Z ES footprint. In the FR4, the T_s point is further away due to the positioning of the vias for optimum thermal resistance. The T_s points for low and high density MCPCB relative to the LUXEON Z ES emitter package are slightly different because of the need to maintain a minimum width (typical min of 150um depending on PCB manufacturer capability) of the solder mask between the T_s and the thermal pad to avoid joining of these two openings during PCB manufacturing.

The thermal resistance $R\theta_{j\text{-}s}$ between the LUXEON Z ES junction and T_s point for single LUXEON Z ES emitter was experimentally determined and shown in Table 3.

Table 3. Typical thermal resistance between junction and T_s point.

BOARD TYPE (SEE SECTION 3.1 AND 3.2 FOR DETAIL BOARD PARAMETERS USED	$R\theta_{j\text{-}s}$ (K/W)
1mm FR4 open PTH vias (1oz + 1oz for additional plating)	9.0
1mm 1-up MCPCB 1oz copper (design B)	7.0
1mm 1-up MCPCB 2oz copper (design B)	6.0
1mm 1-up MCPCB 1oz copper (design A)	8.0
1mm 1-up MCPCB 2oz copper (design A)	7.0
1mm 4-up MCPCB 1oz copper (design B)	2.0
1mm 4-up MCPCB 2oz copper (design B)	1.5
1mm 9-up MCPCB 1oz copper (design B)	0.9
1mm 9-up MCPCB 2oz copper (design B)	0.5

The junction temperature can then be calculated as follows:

$$T_j = T_s + R\theta_{j\text{-}s} \cdot P_{electrical}$$

In this equation $P_{electrical\_array}$ is the electrical power going into the LUXEON Z ES emitter. The $R\theta_{j\text{-}s}$ result is only valid for the board parameters as described in section 3.1 and 3.2.

LED board configurations with multiple closely packed LUXEON Z ES emitters may require additional thermal modeling or measurements to determine the pad temperature, especially for those LUXEON Z ES emitters which are in the center of an array and are not easily accessible.

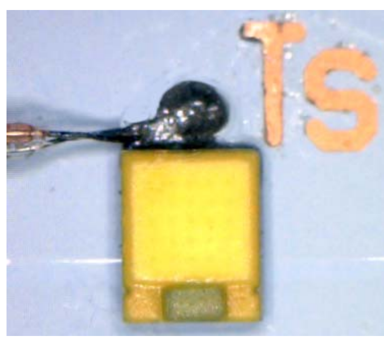
5. Solder Reflow Guidelines

5.1 Stencil, Solder Mask Design, and Silk Screen Labels

Given the small size of the electrical and thermal pads of the LUXEON Z ES emitter, it is important that the appropriate amount of solder paste is dispensed onto the PCB prior to reflow of LUXEON Z ES emitters. The recommended solder mask and stencil design for the LUXEON emitter is included in the PCB footprint design of Figure 6 and Figure 7. The recommended stencil thickness is 100µm or 4 mils for all the designs. Note that any silk screen labels (anode/cathode markers, LED #, etc.) on top of the solder mask should be placed at least 2 mm outside the outline of the LUXEON Z ES emitter. If labels are placed too close to the solder mask openings, the height of the ink may interfere with the stencil paste printing quality. Alternatively, use a copper or FR4 defined text but need to ensure that the solder mask width is sufficient to avoid missing solder mask area during PCB manufacturing. Typically most PCB manufacturers can process 150um solder mask width but please consult with each manufacturer. With copper defined text, consideration on electrical safety for approbation needs to be considered since the opening, if big enough can be made accessible to human touching.

The actual volume and placement of solder paste onto the PCB has a direct impact on the solder joint quality and reliability after reflow. Too little solder paste may result in poor solder coverage after reflow while too much solder paste may cause the LUXEON emitter to tilt, rotate and/or cause solder bridging during placement and reflow. In addition, poor placement of the stencil prior to solder dispense may result in some of the solder paste being screen printed onto the solder mask, reducing the amount of solder paste which is available to establish a strong solder joint during reflow.

In general if the solder and stencil pattern mask opening is small, higher tolerance and assembly process accuracy is required and vice versa. This will limit the choice of a manual versus automatic assembly process. More information can be found in section 5.3. Figure 15 to Figure 17 show examples of board layout issues, solder reflow and stencil printing issues.



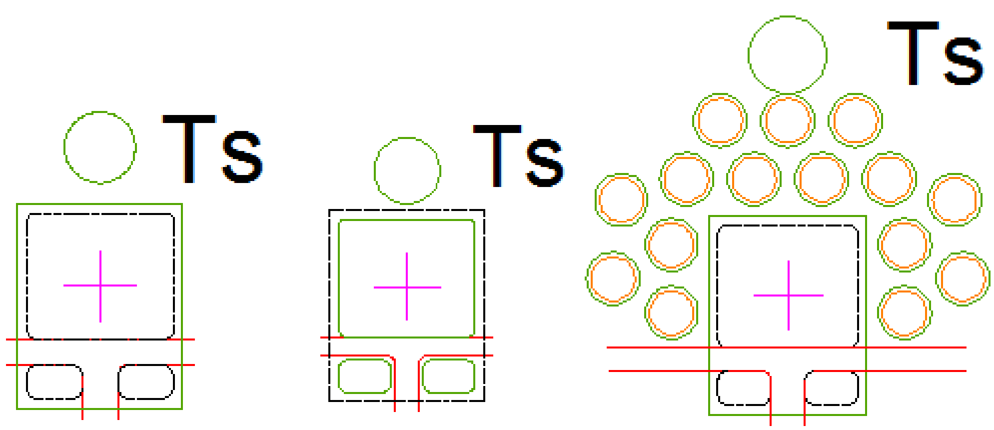


Figure 14. An example of the location of T_s point for MCPCB low density, MCPCB high density and FR4 (left to right) and an actual thermocouple attached to T_s point on Design B (top).

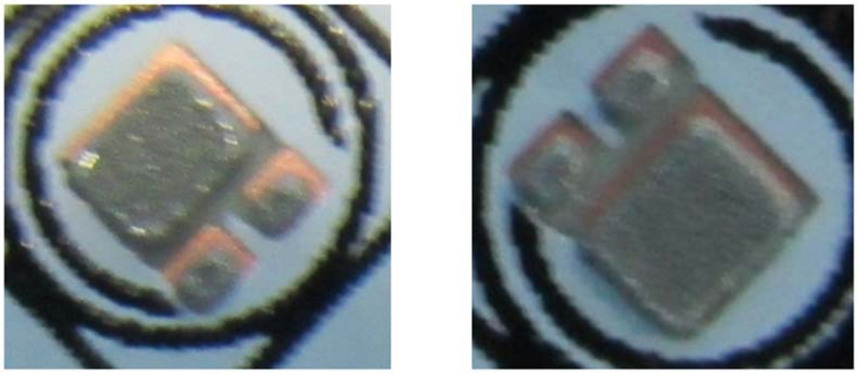


Figure 15. A poorly placed stencil may cause some of the solder paste to be deposited onto the solder mask, reducing the amount of solder paste which is available on the pads to establish a strong solder joint. In this example, the silk screened circles within the LUXEON package outline may also lift the stencil from the solder mask, compromising the quality of the solder paste deposition process.

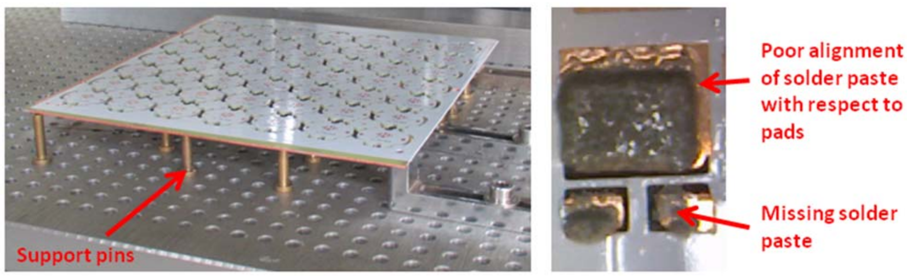


Figure 16. PCB panels should be rigidly supported during solder paste printing to ensure proper alignment between the stencil and the PCB as well as reliable transfer of solder paste onto the PCB. A rigid support panel is preferred over multiple support pins, especially for PCB panels with v-scores or perforated holes for de-panel purposes.

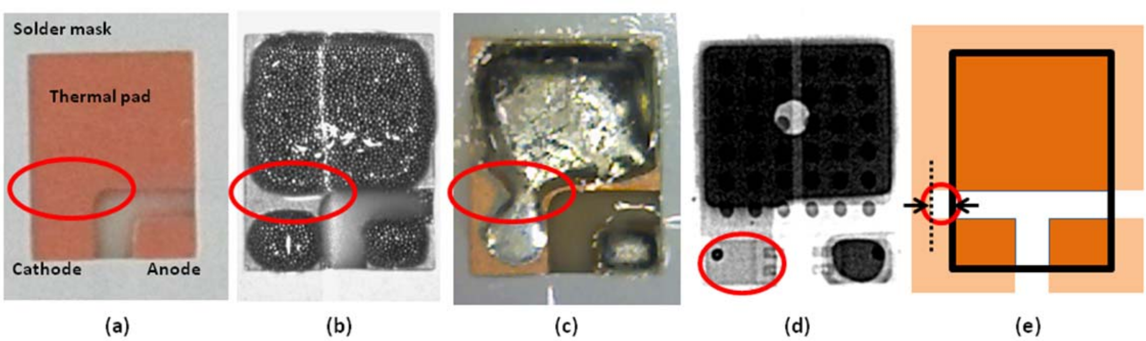


Figure 17. An incorrectly laid out PCB may result in solder being wicked away from the smaller electrical pad to the larger neighboring thermal pad. In this example, the cathode is directly connected to the thermal pad without any solder mask barrier in between (a). Even though the solder paste on the thermal pad and cathode is disjointed after dispense (b), during reflow the solder bridges between the cathode and thermal pad (c). In the worst case scenario, all the solder is wicked away from the smaller cathode to the larger thermal pad causing an electrical open (d). The PCB should, therefore, be designed such that any connection between the electrical pad and thermal pad remains covered by the solder mask even for the worst possible alignment error between the solder mask and the metallization (e).

5.2 Solder Paste

Lumileds recommends using a lead-free solder paste for LUXEON emitters. Lumileds successfully tested a grade 3 solder paste with satisfactory results. For example Alpha Metal SAC305-CVP390-M20 type 3. However, since application environments vary widely, Lumileds recommends that customers perform their own solder paste evaluation in order to ensure it is suitable for the targeted application.

5.3 Solder Paste Screen Printing

1. The stencil is manually aligned to the PCB prior to printing. No adjustments are made during printing.
2. The stencil is manually aligned to the PCB prior to printing. During printing, the machine keeps track of the PCB fiducial mark(s) and makes any necessary adjustments to maintain proper alignment with the PCB.
3. A technician performs a crude alignment of the stencil to the PCB. During printing, the machine keeps track of the PCB fiducial mark(s) and the stencil fiducial mark(s) and maintains proper alignment between the fiducials throughout the process.

Method 1 has the worst accuracy and repeatability of the three methods discussed. Method 2 offers the same accuracy as method 1 but ensures better repeatability. Method 3 has the best accuracy and best repeatability of the 3 methods discussed.

Depending on what screen printing method is used, the size of the anode and cathode solder mask openings on the PCB may have to be enlarged to compensate for any misalignments between the stencil and the PCB panel. The size of the anode and cathode openings in the stencil should be enlarged accordingly. Given the large size of the thermal pad compared to the anode and cathode pads of the LUXEON emitter, the size of the thermal pad typically does not require any modifications in the solder mask or stencil. Note, though, that any changes in the solder mask opening for anode and cathode pads should not change the spacing between the three pads on the PCB, i.e. the spacing between the two electrical pads should be 0.35mm while the spacing between the electrical pads and the thermal pad should be 0.25mm.

In order to ensure proper alignment between the stencil and the PCB as well as reliable transfer of solder paste onto the PCB, all PCB panels should be rigidly supported during solder paste printing. Instead of placing the PCB panel on multiple support pins (see Figure 16), it is best to place the PCB panel on a single solid plate. This is particularly important for PCB panels which contain v-scores or perforated holes for de-panel purposes.

5.4 Solder Reflow Profile

The LUXEON emitter is compatible with standard surface-mount and lead-free reflow technologies. This greatly simplifies the manufacturing process by eliminating the need for adhesives and epoxies. The reflow step itself is the most critical step in the reflow soldering process and occurs when the boards move through the oven and the solder paste melts, forming the solder joints. To form good solder joints, the time and temperature profile throughout the reflow process must be well maintained.

A temperature profile consists of three primary phases:

1. Preheat: the board enters the reflow oven and is warmed up to a temperature lower than the melting point of the solder alloy.
2. Reflow: the board is heated to a peak temperature above the melting point of the solder, but below the temperature that would damage the components or the board.
3. Cool down: the board is cooled down, allowing the solder to freeze, before the board exits the oven.

For detailed information on the recommended reflow profile, refer to the IPC/JEDEC J-STD-020C reflow profile in the appropriate datasheet for each LUXEON product.

5.5 Placement Accuracy

In order to achieve the highest placement accuracy Lumileds recommends using an automated pick and place tool with a vision system that can recognize the bottom metallization or package outline of the LUXEON emitter (see Section 6 for more details).

5.6 Solder Wetting and Voids

To ensure good solder joint reliability, the solder reflow process should be tuned such that dewetting and solder voids after reflow are minimized. Lumileds recommends a maximum specification of 25% on the combined dewetting and solder void area under critical pads. However, since application environments vary widely, customers should always perform their own evaluation in order to ensure that the maximum allowable amount of dewetting and solder voids is suitable for the targeted application and operating conditions.

If excessive dewetting is observed, standards JESD22-B102E and IPC J-STD-003 provide guidelines on how to assess the solderability of the surface mount component and the corresponding PCB, respectively. According to these specifications a minimum of 95% of the critical surfaces tested shall exhibit good wetting. For LUXEON emitters the critical area is typically defined as the area of the pads on the LED.

6. Assembly Process Guidelines

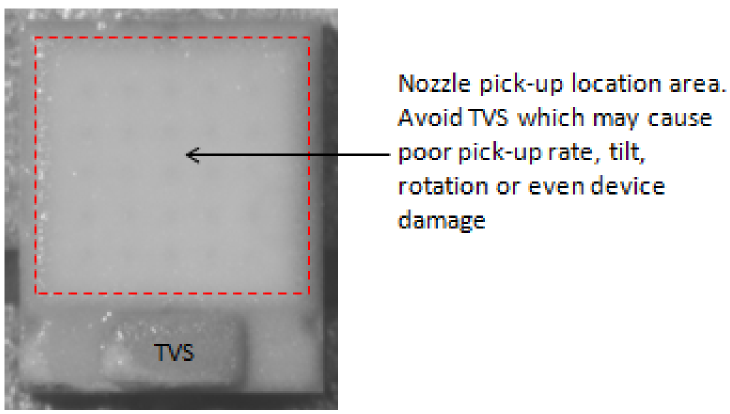


Figure 18. Nozzle pick-up location area for LUXEON Z ES emitters.

6.1 Pick-and-Place

Automated pick and place equipment provides the best handling and placement accuracy for LUXEON Z ES emitters. Figure 19 to Figure 22 show various pick and place nozzle designs and corresponding machine settings which have been successfully used to pick and place LUXEON Z ES emitters with pick and place equipment from Juki, Panasonic, Yamaha and Samsung.

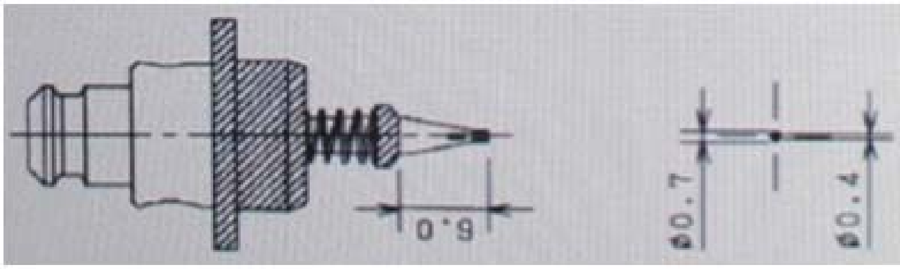
Note that pick and place nozzles are customer specific and are typically machined to fit specific pick and place tools. Based on these pick and place experiments. Lumileds advises customer to take the following general pick and place guidelines into account:

a. The tip of the nozzle should be positioned on the flat surface above the LED chip area; the area around the TVS should be avoided because of height differences (see Figure 18).

b. The nozzle tip should be clean and free of any particles since this may interact with the silicone coating of the LUXEON Z ES during pick and place.

c. During setup and the first initial production runs, it is a good practice to inspect the top surface of LUXEON Z ES emitters under a microscope to ensure that emitters are not accidentally damaged by the pick and place nozzle.

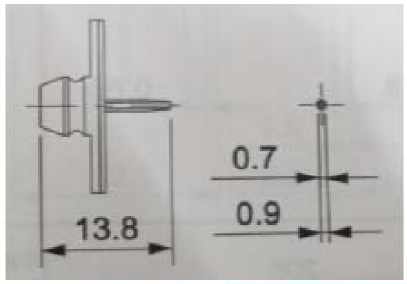
Since LUXEON Z ES has no primary optics or lens which can act as a mechanical enclosure protection for the LED chips, the pick-up and placement force applied to the top of the package should be kept to minimum. The pick-up area on LUXEON Z ES is defined in Figure 18. The placement force (consisting of impact force and dwell force, also known as static force) depends on the nozzle tip material, nozzle spring stiffness, nozzle diameter, vacuum pressure, over travel distance, PCB height differences and PCB warping.



PICK AND MOUNT INFORMATION	
XY	Fast2
Pick depth	0mm
Picking stroke	0mm
Pick Z down	Fast2
Pick Z up	Fast2
Placing stroke	0mm
Place Z down	Fast2
Place Z up	Fast2
Theta (Measure)	Fast
Theta (Other)	Fast

VISION INFORMATION	
Centering method	Laser
Comp shape	Corner Square

Figure 19. Pick and place nozzle design and machine settings for Juki KE2080L in combination with the off-the-shelf nozzle "502." All dimensions in mm.



PICK AND MOUNT INFORMATION	
Gap – Mount	0mm
Gap – Pick	0mm
Pickup position – Z	-0.16mm
Fdr drive time	Std
Pickup keep time	Std
Mount keep time	Std
Pickup speed	80
Mount speed	60

VISION INFORMATION	
Ref	51
Recognition speed	Mid
Recognition height	0mm
Lamp 1	0
Lamp 2	0
Lamp 3	30
Lamp 4	0
Lamp 5	0
Lamp 6	0
Lamp 7	0
Lamp 8	0

Figure 20. Pick and place nozzle design and corresponding machine settings for Panasonic CM402 in combination with the off-the-shelf nozzle "115."

PICK AND MOUNT INFORMATION	
Pick timer	0s
Mount timer	0s
Pick height	0.16mm
Mount height	0mm
Mount action	Normal
Mount speed	100%
Pickup speed	100%
Vacuum check	Normal Chk
Pick vacuum	20%
Mount vacuum	60%

VISION INFORMATION	
Alignment group	Special
Alignment type	Odd. Chip
Alignment module	Fore & Back & Las
Light selection	Main + Coax
Lighting level	6/8
Comp. threshold	60
Comp. tolerance	15
Search area	1.5mm
Comp. intensity	N.A.
Auto threshold	Not Used

Figure 21. Pick and place nozzle design and corresponding machine settings for Yamaha YV100X in combination with the off-the-shelf nozzle "7WA."

CN040 Dimensions
Length = 13.50mm
Inner nozzle diameter = 0.38mm

PICK AND MOUNT INFORMATION	
Pick Height	0.16mm
Mount Height	0mm
Delay – Pick Up	30msec
Delay – Place	40msec
Delay – Vacuum Off	0msec
Delay – Blow On	0msec
Speed – XY	1
Speed – Z Pick Down	1
Speed – Z Pick Up	1
Speed – R	1
Speed – Z Place Down	1
Speed – Z Place Up	1
Z Align Speed	1
Soft Touch	Not use
Mount Method	Normal

VISION INFORMATION	
Camera No	Fly Cam2
Side	13
Outer	9

Figure 22. Pick and place nozzle design and corresponding machine settings for Samsung SM421 in combination with the off-the-shelf nozzle "CN040."

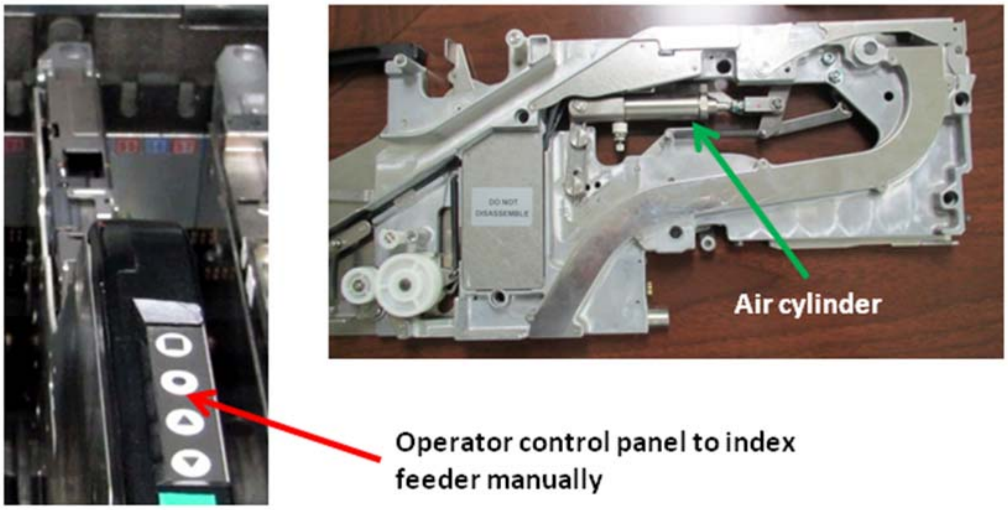


Figure 23. Examples of an electric feeder (left) and a pneumatic feeder (right) which are typically used in pick and place machines to advance the tape with LEDs.

6.2 Pick-and-Place Machine Optimization

Pick and place machines are typically equipped with special pneumatic or electric feeders to advance the tape containing the LEDs. In pneumatic feeders, air pressure is used to actuate an air cylinder which then turns the sprocket wheel to index the pocket tape; electric feeders, in contrast, use electric motors to turn the sprocket wheel (see Figure 23). Electric feeders often also contain a panel which allows an operator to control the electric feeder manually.

The indexing step in the pick and place process may cause some LEDs to accidentally jump out of the pocket tape or may cause some LEDs to get misaligned inside the pocket tape, resulting in pick-up errors. Depending on the feeder design, minor modifications to the feeder can substantially improve the overall pick and place performance of the machine.

For information, there are many types of pick and place feeder designs available. Some feeders can be used as-is without any further modifications, some feeders require a shift in the position where the cover tape is peeled off the tape, and yet other feeders require the shutter to be completely removed so that the cover tape peeling position can be adjusted. Figure 24 shows representative pictures of each feeder design when used on domed parts as an example (not applicable to LUXEON Z ES). Since there are many different feeder designs in use, it is important to understand the basic principle behind modifying the feeders so that effective modifications can still be carried out when different feeder designs are encountered. Note that for LUXEON Z ES pick and place, there is no need to remove the shutter or shift the cover tape peeling location unless one encounters a problem and trying to resolve pick-up issues.

To minimize the jerking of components in pneumatic feeders during indexing, it may be necessary to install an air pressure control valve. In some pneumatic feeder designs, such a control valve is already integrated by the machine supplier; in others an external control valve may have to be installed (see Figure 25).

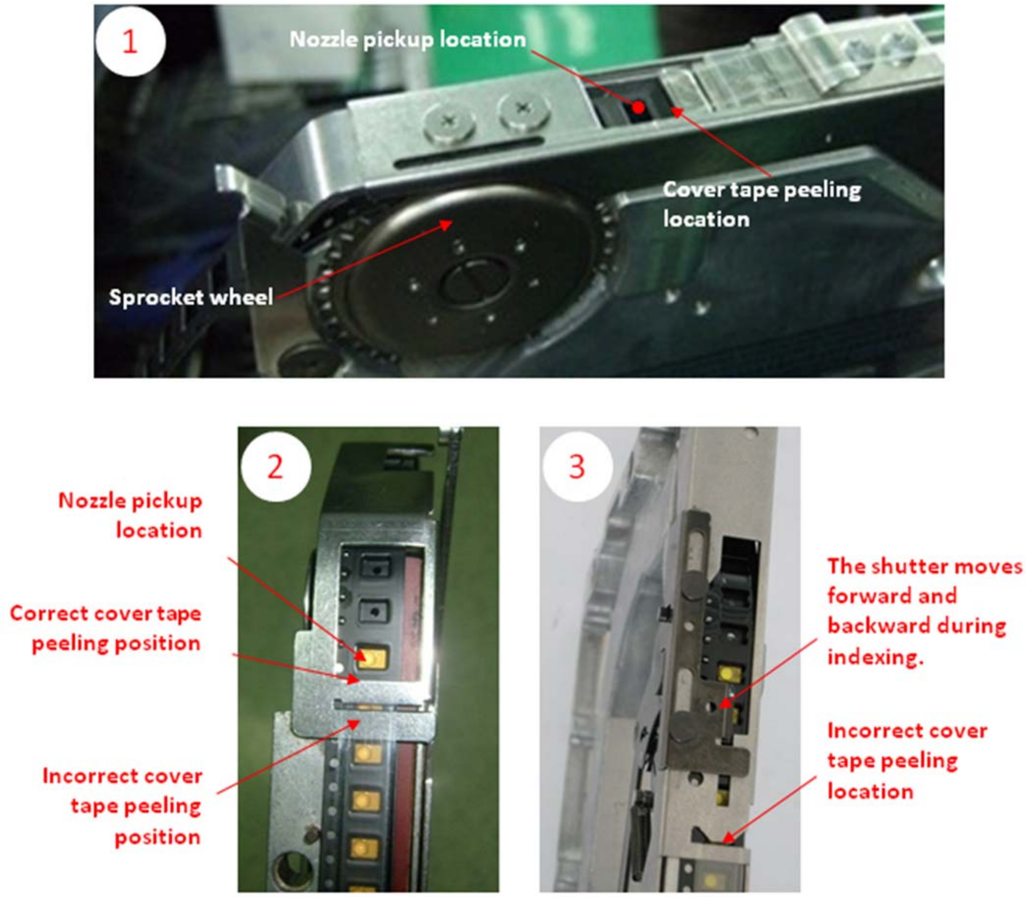


Figure 24. Not applicable for LUXEON Z ES but to illustrate different feeder designs and its suitability for domed parts. Feeder 1 does not require any modification. Feeder 2 requires the cover tape peeling position to be shifted. Feeder 3 requires the shutter to be removed before the cover tape peeling position can be adjusted.

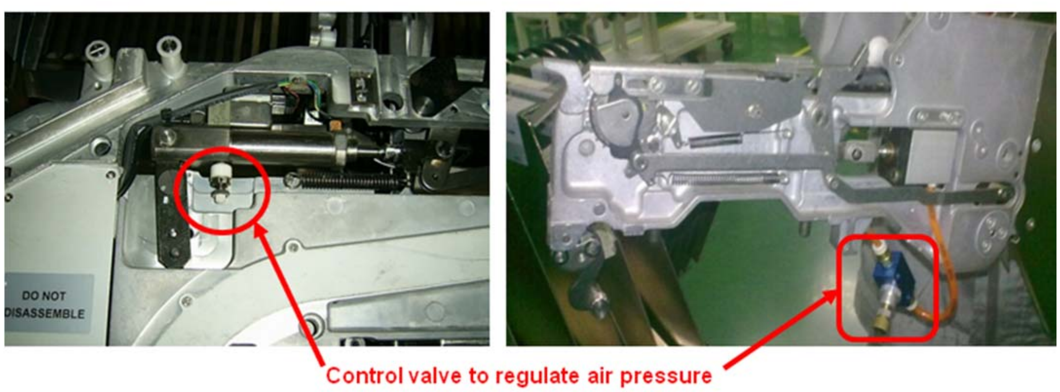


Figure 25. Pneumatic feeder with integrated air pressure control valve (left) and pneumatic feeder with air pressure control valve installed afterwards (right).

Figure 26. PCBs are typically shipped in moisture proof packaging with desiccant and a humidity indicator.

7. PCB Inspection and Handling Guidelines

7.1 Introduction

Given the small footprint of the LUXEON Z ES emitter, it is important that all PCBs are handled according to industry standards to ensure solderability of the LUXEON emitters onto the PCBs. In particular, to avoid contamination of PCBs and to prevent PCBs from absorbing moisture during delivery, receiving, stocking, assembly and soldering, PCBs should be stored and handled per the guidelines spelled out in industry standard IPC-1601 "Printed Board Handling and Storage Guidelines."

7.2 Packaging

PCBs are typically shipped in moisture proof packaging with desiccant and a humidity indicator card, which changes color (typically from blue to pick) with increasing humidity (see Figure 26). The desiccant absorbs any moisture that may enter the bag and the humidity indicator card will provide an easy visual indication of the moisture level should there be an exposure. If the 10% dot on the humidity indicator card changes color the moisture proof packaging of the PCBs is most likely compromised. In those situations, the PCBs should be baked before use.

If PCBs are exposed to a factory ambient environment (i.e. less than 30°C/60% RH) for less than 30 minutes, the PCBs can be re-packed with the original moisture barrier bag using a vacuum sealing machine. If the exposure to a factory ambient environment does not exceed 60 minutes, the PCBs can be returned to a dry storage cabinet with a relative humidity of at most 10%. If PCBs are exposed to a condition not fulfilling the above requirements, then the PCBs should be baked before use. The appropriate bake time and temperature depends on the surface finish of the PCB as outlined in IPC-1601 "Printed Board Handling and Storage Guidelines."

With regards to cleanliness of incoming PCBs, ionic contamination should be kept below the maximum limit of 1.56μg NaClEq./cm^2. This is in line with the guidelines spelled out in IPC-6012 "Qualification and Performance Specification for Rigid Printed Boards."

7.3 Inspection of Incoming PCBs

To inspect the quality of incoming PCBs, it is best to adopt the inspection criteria in IPC-A-600F "Acceptability of Printed Boards." Figure 27 shows examples of unacceptable defects and contamination on incoming PCBs.

During PCB manufacturing poor positional control of the solder mask layer with respect to the top metallization can impact the pad dimensions on the PCB. Figure 28 shows four top-view pictures of the solder mask opening and underlying copper layer on the same 4-up PCB. In this particular example, the anode and cathode dimensions for three out of the four LUXEON emitters are smaller than designed. Consequently, some solder paste may be placed onto the solder mask during screen printing, causing the electrodes to have insufficient solder paste to make a reliable electrical connection.

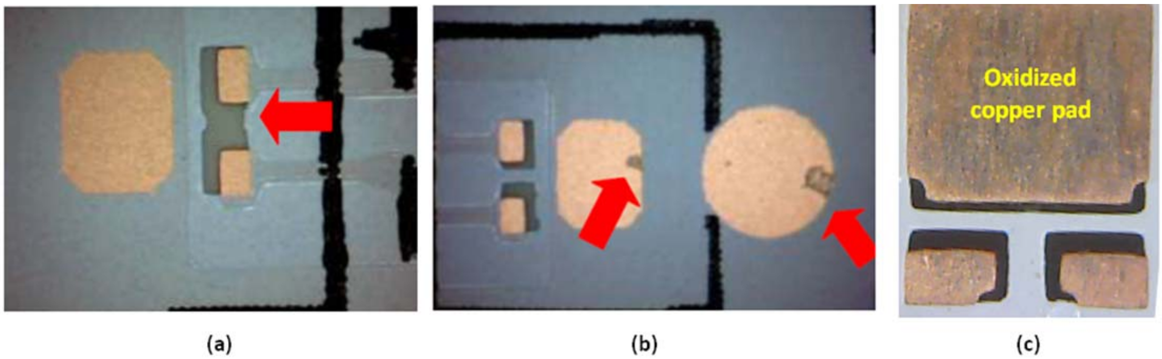


Figure 27. Examples of unacceptable defects and or contamination on incoming PCBs: missing solder resist between electrical pads (a), particles on exposed copper pads (b), and oxidized copper pads possibly due to poor OSP control, which may cause de-wetting and/or non-wetting of the pads during reflow (c).

Figure 28. During PCB manufacturing poor positional control of the solder mask layer with respect to the top metallization layer can impact the actual pad dimensions on the PCB. The four pictures in this example were all taken from the same physical 4-up PCB. The anode and cathode dimensions for positions (a) – (c) on this PCB are smaller than originally designed. Only the pad dimensions for position (d) were in specification.

Figure 29 shows examples of poor PCB workmanship, possibly due the solder mask strip process. These can cause problems when depositing solder paste onto the pads and making electrical connection when the LUXEON emitter is placed onto the PCB.

Figure 29. Examples of poor PCB workmanship which may adversely impact the solder joint between the PCB and the LUXEON emitter.

8. Packaging Considerations — Chemical Compatibility

The LUXEON Z ES package contains a silicone overcoat to protect the LED chip. As with most silicones used in LED optics, care must be taken to prevent any incompatible chemicals from directly or indirectly reacting with the silicone.

The silicone overcoat in LUXEON Z ES is gas permeable. Consequently, oxygen and volatile organic compound (VOC) gas molecules can diffuse into the silicone overcoat. VOCs may originate from adhesives, solder fluxes, conformal coating materials, potting materials and even some of the inks that are used to print the PCBs.

Some VOCs and chemicals react with silicone and produce discoloration and surface damage. Other VOCs do not chemically react with the silicone material directly but diffuse into the silicone and oxidize during the presence of heat or light. Regardless of the physical mechanism, both cases may affect the total LED light output. Since silicone permeability increases with temperature, more VOCs may diffuse into and/or evaporate out from the silicone.

Careful consideration must be given to whether LUXEON Z ES emitters are enclosed in an “air tight” environment or not. In an “air tight” environment, some VOCs that were introduced during assembly may permeate and remain in the silicone overcoat. Under heat and “blue” light, the VOCs inside the silicone overcoat may partially oxidize and create a silicone discoloration, particularly on the surface of the LED where the flux energy is the highest. In an air rich or “open” air environment, VOCs have a chance to leave the area (driven by the normal air flow). Transferring the devices which were discolored in the enclosed environment back to “open” air may allow the oxidized VOCs to diffuse out of the silicone overcoat and may restore the original optical properties of the LED.

Determining suitable threshold limits for the presence of VOCs is very difficult since these limits depend on the type of enclosure used to house the LEDs and the operating temperatures. Also, some VOCs can photo-degrade over time.

Table 4 provides a list of commonly used chemicals that should be avoided as they may react with the silicone material. Note that Lumileds does not warrant that this list is exhaustive since it is impossible to determine all chemicals that may affect LED performance.

The chemicals in Table 4 are typically not directly used in the final products that are built around LUXEON Z ES LEDs. However, some of these chemicals may be used in intermediate manufacturing steps (e.g. cleaning agents).

Consequently, trace amounts of these chemicals may remain on (sub) components, such as heat sinks. Lumileds, therefore, recommends the following precautions when designing your application:

- When designing secondary lenses to be used over an LED, provide a sufficiently large air-pocket and allow for “ventilation” of this air away from the immediate vicinity of the LED.
- Use mechanical means of attaching lenses and circuit boards as much as possible. When using adhesives, potting compounds and coatings, carefully analyze its material composition and do thorough testing of the entire fixture under High Temperature over Life (HTOL) conditions.

Table 4. List of commonly used chemicals that will damage the silicone overcoat of LUXEON Z ES. Avoid using any of these chemicals in the housing that contains the LED package.

CHEMICAL NAME	NORMALLY USED AS
Hydrochloric acid	acid
Sulfuric acid	acid
Nitric acid	acid
Acetic acid	acid
Sodium hydroxide	alkali
Potassium hydroxide	alkali
Ammonia	alkali
MEK (Methyl Ethyl Ketone)	solvent
MIBK (Methyl Isobutyl Ketone)	solvent
Toluene	solvent
Xylene	solvent
Benzene	solvent
Gasoline	solvent
Mineral spirits	solvent
Dichloromethane	solvent
Tetracholorometane	solvent
Castor oil	oil
Lard	oil
Linseed oil	oil
Petroleum	oil
Silicone oil	oil
Halogenated hydrocarbons (containing F, Cl, Br elements)	misc
Rosin flux	solder flux
Acrylic tape	adhesive

RoHS COMPLIANT

About Lumileds

Lumileds is the light engine leader, delivering innovation, quality and reliability.

For 100 years, Lumileds commitment to innovation has helped customers pioneer breakthrough products in the automotive, consumer and illumination markets.

Lumileds is shaping the future of light with our LEDs and automotive lamps, and helping our customers illuminate how people see the world around them.

To learn more about our portfolio of light engines, visit lumileds.com.

LUMILEDS

lumileds.com