APPLICATION NOTE

Non-Isolated DC/DC Converter
Eutectic Solder Process

Introduction

This document provides information on pad sizes, stencil aperture, solder paste and soldering profile for d-PWER™ POL Converters.

STENCIL DESIGN FOR PERIMETER PADS

The stencil aperture is designed to match the land area on the PCB. It is recommended to use a pad to aperture aspect ratio of 90% to 100% with a recommended stencil thickness of 0.125mm.

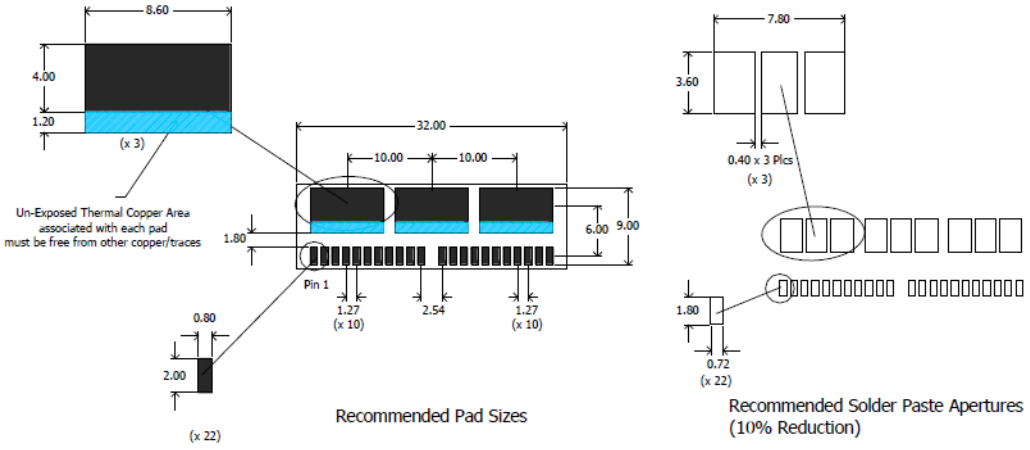

Figure 1. Recommended pad layout for DP71XX Vertical Units

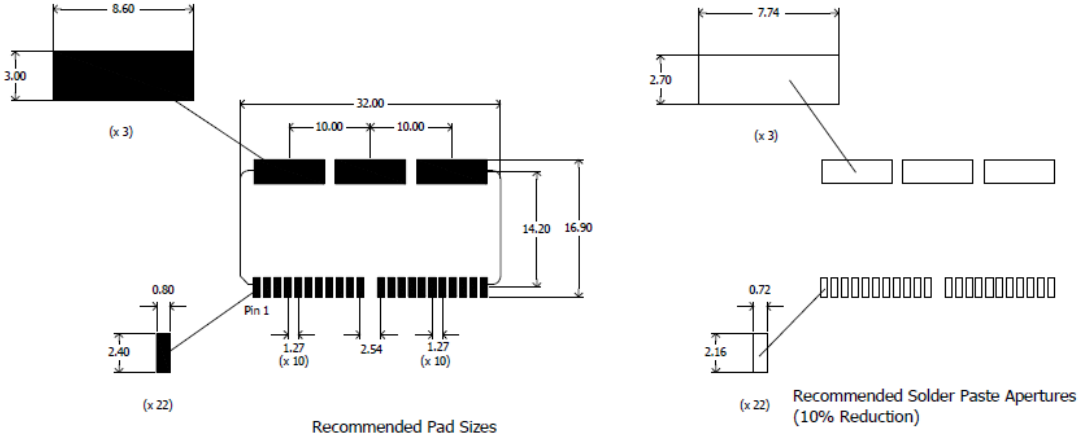

Figure 2. Recommended pad layout for DP70XX Horizontal Units

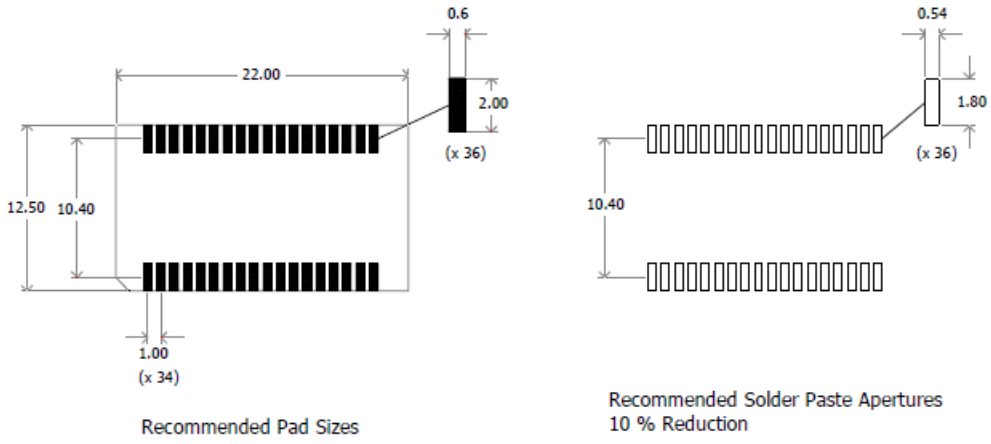

Figure 3. Recommended pad layout for DP7007 Units

Stencil design for thermal pads

In order to effectively remove the heat from the module and to enhance electrical performance the thermal pads need to be soldered to the host PCB with a minimum level of voiding. Eliminating voids may not be possible due to the presence of thermal vias, and also due to the large size of the thermal pads. If the solder paste coverage is too big then out gassing can cause defects such as solder balling and splatter during reflow. To solve these potential issues it is recommended to use smaller multiple openings instead of one bigger opening to print paste in the thermal pad region.

Stencil apertures design

Stencil parameters such as aperture area ratio and stencil fabrication have a significant effect on the volume of paste deposited onto the host PCB. Nickel-plated electro-polished chemically etched or laser-cut stencils are recommended. A 5o taper on the aperture while not necessary is also recommended to facilitate paste release. For optimal reliability and thermal performance Bel Power Solutions recommends the 90 to 100% pad to aperture ratio.

Solder paste

Solder paste inspection is a key process step to ensuring a good consistent reliable joint. Paste volume is the best indicator for this. The use of stainless steel stencils and stainless steel blades is recommended to ensure consistent paste deposit. The recommended stencil thickness is 0.005” (0.125 mm).
It is also recommended to use a no-clean type 3 solder paste.

Soldering and reflow profiles

The d-PWER™ POL Converters is a surface-mountable module, thereby electrical and thermal connection to the host PCB is made by printing solder paste onto the host PCB, placing the component, and reflowing the solder using a convection or infrared oven. As with all components, the actual temperature of the pins is a function of the host PCB thickness, component density, copper weight and surrounding component size.
Note that in the case of infrared ovens, particular attention must be paid to the location of large devices as the placement of a large device close to the d-PWER™ POL Converters can cause a shadowing effect. Figure 4 shows a typical reflow profile. The ramp rate during pre-heat should not exceed 3oC/second; the time above liquids should range between 60 and 100 seconds depending on the host PCB thickness.

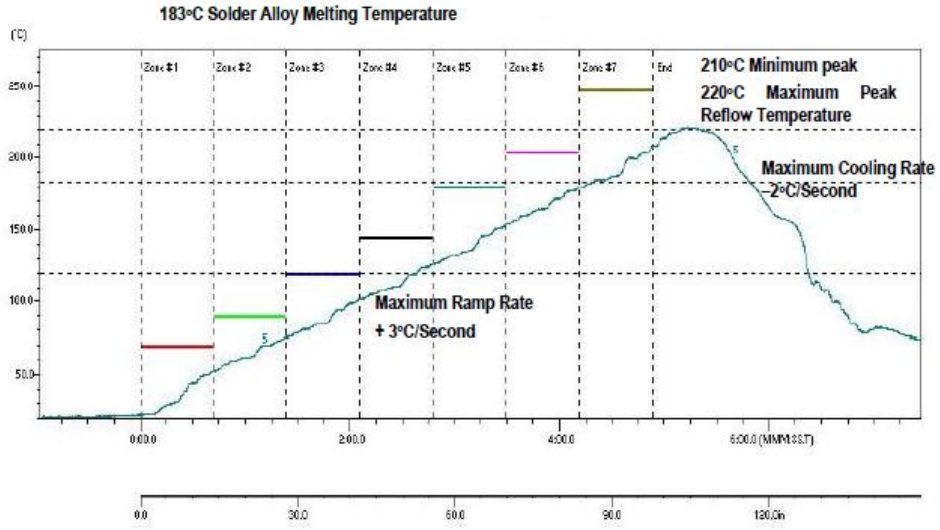

Figure 4. Typical reflow profile

d-PWER™ POL CONVERTERS REMOVAL

Equipment required

PDR Infrared Rework Unit

Materials

Flux

Procedure summary

The procedure outlined below is generic in nature and identifies the procedural steps which need to be accomplished to effect d-PWER™ POL Converters removal. Each step must be tailored to accommodate the attributes and characteristics of the specific system being used (system manufacturers usually provide generalized operating procedures which must be further refined to achieve optimum results).

Procedural preconditions

The following precondition shall be accomplished prior to performing the procedure:
Develop a Time/Temperature Profile (TTP) for the specific PCB (see section 1.9, process goals and guidelines in IPC-7711)

Procedure steps

- Clean PCB as appropriate with solvent.
- Place the PCB in the system workpiece holder.
- Inject liquid flux onto the pads.
- Set PDR rework system to achieve the TTP defined by procedural analysis.
- Position the lens so as to be above the center of the back heater.
- Place the PCB under the lens unit, position the PCB and adjust the spot size so that the spot overlaps the device by 2 to 3mm all around.
- Press START button to initiate preheat/reflow cycle defined by procedural analysis.
- After the device's solder has become molten, lift device carefully with tweezers.

d-PWER™ POL CONVERTERS PLACEMENT

The d-PWER™ POL Converters can be placed using conventional pick and place equipment. The placement accuracy required is ±0.05mm. The d-PWER™ POL Converters will be presented to the machine via Tape and Reel. (See datasheet for tape dimensions and drawings).
The recommended pick-up nozzle size is 8mm.

Site preparation (for manual placement)

Perform site preparation in accordance with procedure 4.1.1 or 4.1.3 per IPC-7711

Equipment required

PDR Infrared Rework Unit

Materials

Solder Paste (no-clean type 3 solder paste)

Procedure summary

The procedure outlined below is generic in nature and identifies the procedural steps that need to be accomplished to effect d-PWER™ POL Converters placement. Each step must be tailored to accommodate the attributes and characteristics of the specific system being used (system manufacturers typically provide generalized operating procedures which must be further refined to achieve optimum results).

Procedural preconditions

The following precondition shall be accomplished prior to performing the procedure.

- Develop a Time/Temperature Profile (TTP) for the specific PCB (see section 1.9, process goals and guidelines in IPC-7711).
- Position the lens so as to be above the center of the back heater.

Procedure steps

- Clean PCB surface and lands
- Apply solder paste (stencil as appropriate).
- Place the PCB in the system workpiece holder.
- Set PDR rework system to achieve the TTP defined by procedural analysis.
- Place device carefully with tweezers, accurately positioning the leads over the pads.
- Place the PCB under the lens unit, position the PCB and adjust the spot size so that the spot overlaps the device by 2 to 3mm all around.
- Press START button to initiate preheat/reflow cycle defined by procedural analysis.
- Perform Visual inspection of PCB.

PDR Infrared rework unit recommended settings:

PARAMETER	DESCRIPTION		
	Top side: Preheat	Top side: Reflow	Bottom side: Preheat
Temperature:	Power: 5.5	Power: 8.5	Ambient to 180ºC
Time:	72 seconds	96 seconds	Through cycle

d-PWER™ POL CONVERTERS REMOVAL PDR METHOD

PROCEDURE

1. Clean PCB as appropriate with solvent.
2. Place the PCB in the system work piece holder. (See Figure 5)
3. Inject liquid flux onto the pads. (See Figure 6)
4. Set PDR rework system to achieve the TTP defined by procedural analysis.
5. Position the lens so as to be above the center of the back heater.
6. Place the PCB under the lens unit, position the PCB and adjust the spot size so that the spot overlaps the device by 2 to 3 mm all around.
7. Press START button to initiate preheat/reflow cycle defined by procedural analysis. (See Figure 7)
8. After the device's solder has become molten, lift device carefully with tweezers. (See Figure 9)

Figure 5: Position PCB

Figure 6: Inject Liquid flux

Figure 7: Press Start

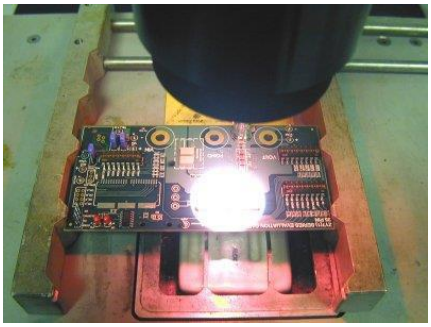

Figure 8: Infra Red heat

Figure 9: Remove Device

d-PWER™ POL CONVERTERS REPLACEMENT PDR METHOD

PROCEDURE

1. Clean PCB as appropriate with solvent.
2. Apply solder paste (See Figure 10)
3. Place the PCB in the system work-piece holder.
4. Set PDR rework system to achieve the TTP defined by procedural analysis.
5. Place device carefully with tweezers, accurately positioning the leads over the pads. (See Figure 12)
6. Place the PCB under the lens unit, position the PCB and adjust the spot size so that the spot overlaps the device by 2 to 3mm all around.
7. Press START button to initiate preheat/reflow cycle defined by procedural analysis. (See Figure 13)
8. Perform Visual inspection of PCB to IPC-610D Standard.

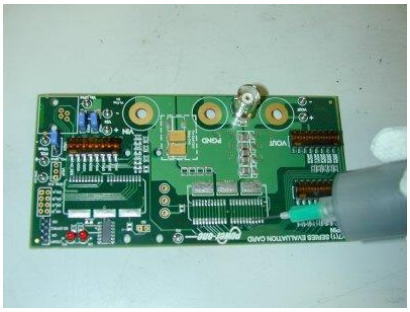

Figure 10: Apply solder paste

Figure 11: Position PCB

Figure 12: Place Unit

Figure 13: Press Start

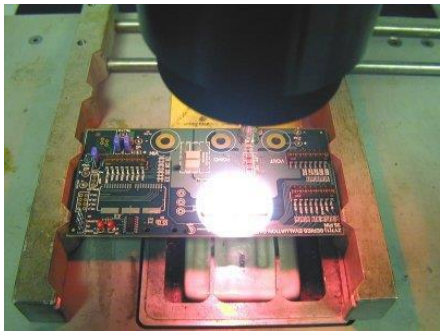

Figure 14: Reflow Unit