Appendix A

Recommended PCB Design Rules for BGA Packages

Xilinx provides the diameter of a land pad on the package side. This information is required prior to the start of the board layout so the board pads can be designed to match the component-side land geometry. The typical values of these land pads are described in Figure A-1 and summarized in Table A-1 for both 0.8 mm and 1.0 mm pitch packages. For Xilinx BGA packages, non-solder mask defined (NSMD) pads on the board are suggested to allow a clearance between the land metal (diameter L) and the solder mask opening (diameter M) as shown in Figure A-1. An example of an NSMD PCB pad solder joint is shown in Figure A-2. It is recommended to have the board land pad diameter with a 1:1 ratio to the package solder mask defined (SMD) pad for improved board level reliability.

The space between the NSMD pad and the solder mask as well as the actual signal trace widths and via dimensions depend on the capability of the PCB vendor. The cost of the PCB is higher when the line width and spaces are smaller.

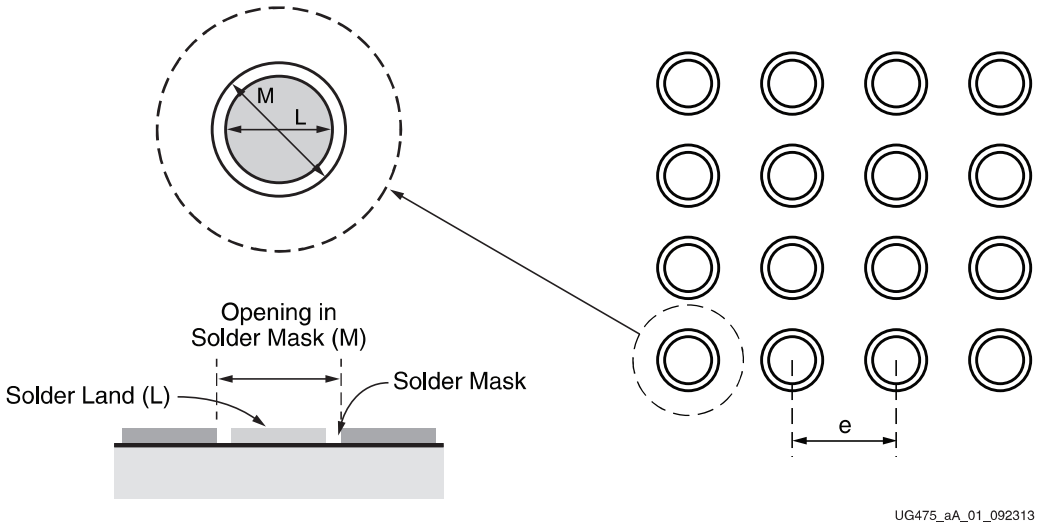

Figure A-1: **Suggested Board Layout of Soldered Pads for BGA Packages**

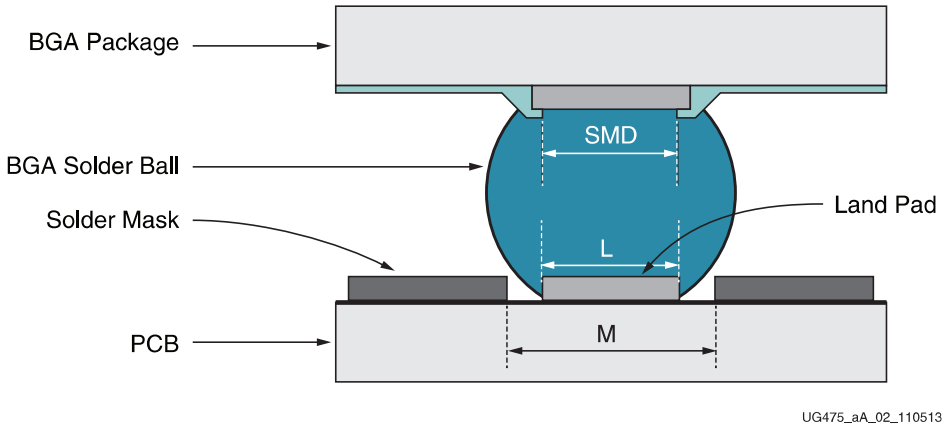

Figure A-2: **Example of an NSMD PCB Pad Solder Joint**

Table A-1: **BGA Package Design Rules**

Packages	0.5 mm Pitch	0.8 mm Pitch	1.0 mm Pitch		
	CPG236	SB/SBG, CS/CSG	FF/FFG, FB/FBG, FH/FHG, FL/FLG, RF/RB/RS	FG/FGG	FT/FTG
Design Rule	**Dimensions in mm (mils)**				
Package land pad opening (SMD)	0.275 mm (10.8 mils)	0.40 mm (15.7 mils)	0.53 mm (20.9 mils)	0.50 mm (19.7 mils)	0.40 mm (15.7 mils)
Maximum PCB solder land (L) diameter	0.275 mm (10.8 mils)	0.40 mm (15.7 mils)	0.53 mm (20.9 mils)	0.50 mm (19.7 mils)	0.40 mm (15.7 mils)
Opening in PCB solder mask (M) diameter	0.375 mm (14.76 mils)	0.50 mm (19.7 mils)	0.63 mm (24.8 mils)	0.60 mm (23.6 mils)	0.50 mm (19.7 mils)
Solder ball land pitch (e)	0.50 mm (19.7 mils)	0.80 mm (31.5 mils)	1.00 mm (39.4 mils)	1.00 mm (39.4 mils)	1.00 mm (39.4 mils)

Notes:

1. Controlling dimension in mm.