

## Recommended PCB Design Rules for BGA and CSP Packages

Xilinx provides the diameter of a land pad on the component 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-2 and summarized in Table A-2.

For Spartan-6 FPGA 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-2. The space between the NSMD pad and the solder mask and the actual signal trace widths depends on the capability of the PCB vendor. The cost of the PCB is higher when the line width and spaces are smaller. In Figure A-2, the 3 x 3 matrix is for illustration only, one land pad shown with via connection.

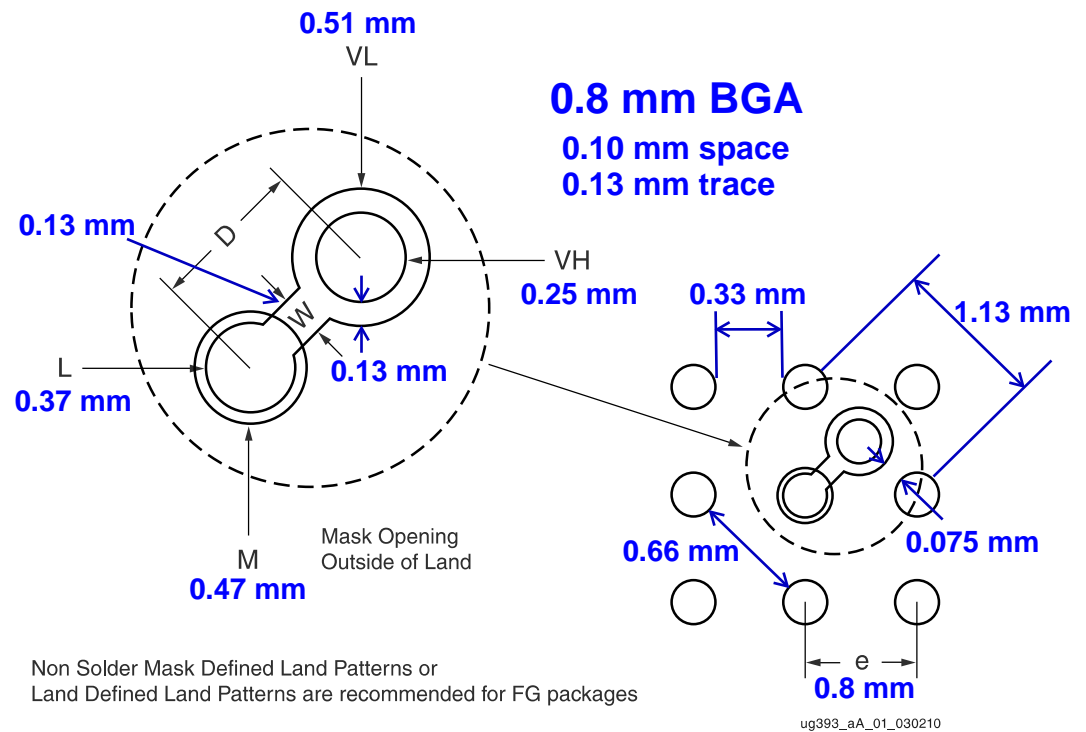


Figure A-2: Suggested Board Layout of Soldered Pads for BGA and CSP Packages