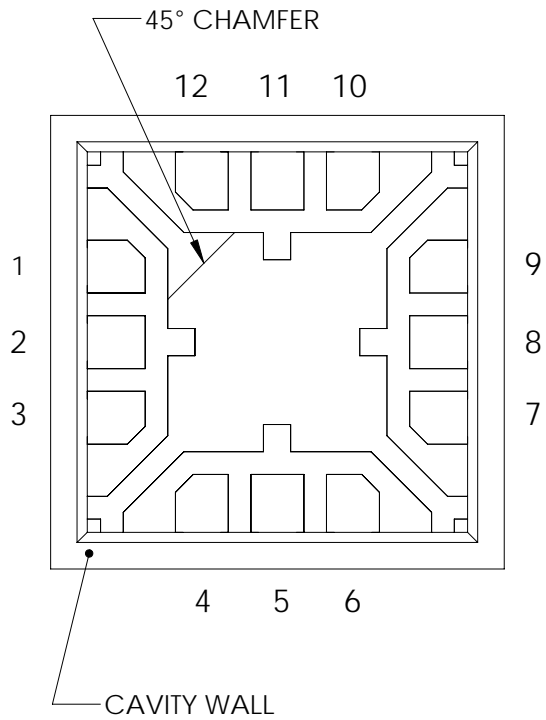
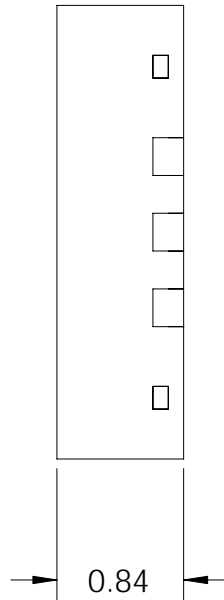


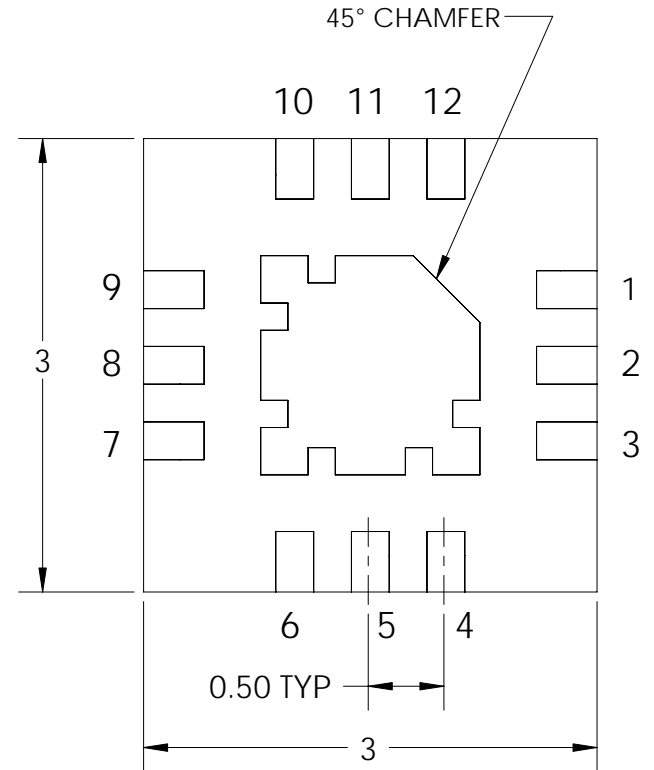
TOP VIEW



SIDE VIEW
(BEFORE LID ATTACH)



BOTTOM VIEW



TITLE:

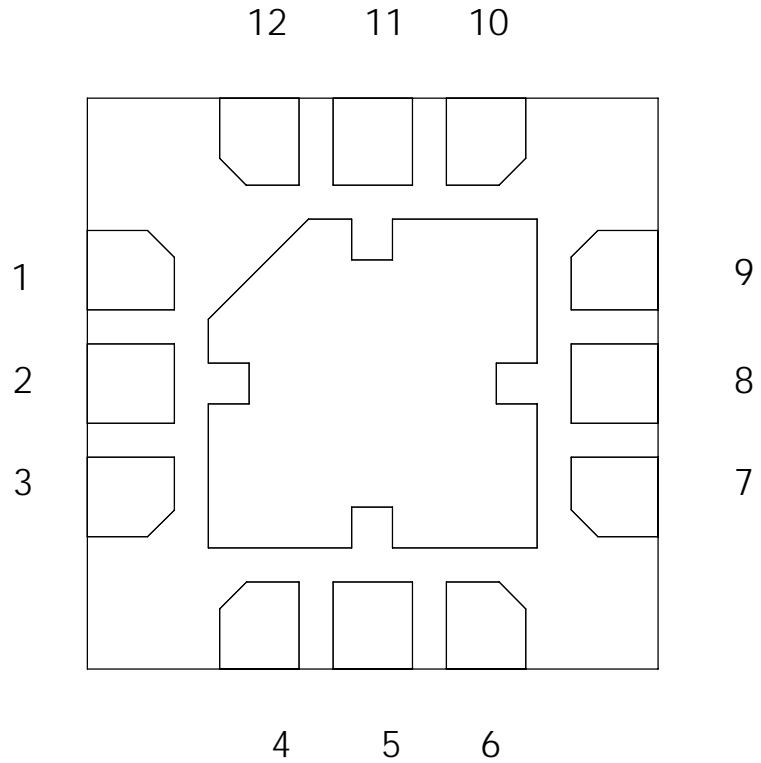
12-LEAD 3mm P=0.5 mm
M-QFN CAVITY PACKAGE

SCALE	SIZE	DWG. NO.	REV
12:1	A	451260 M-QFN12W.5	A

DO NOT SCALE DRAWING

SHEET 2 OF 4

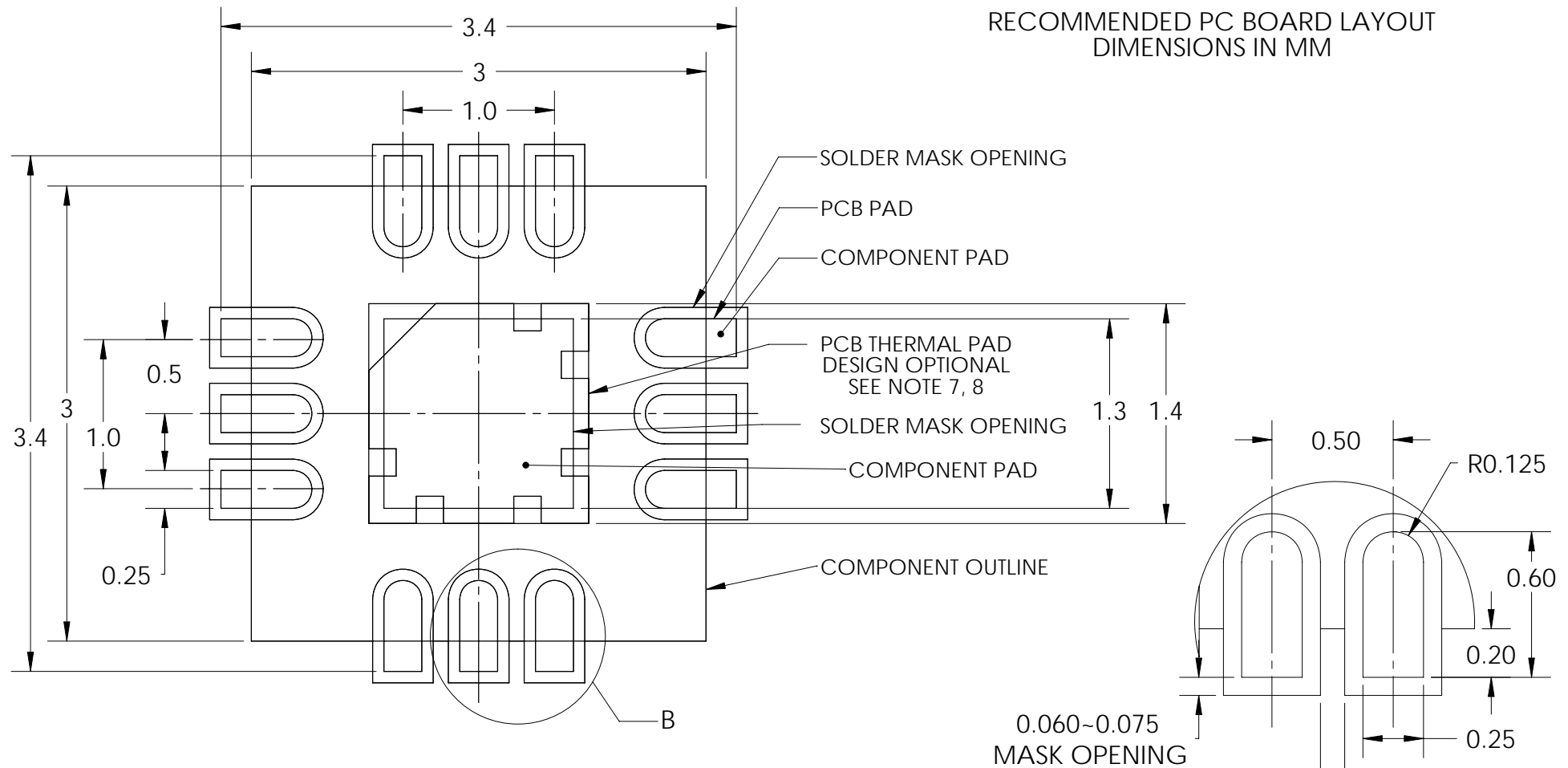
BOND DIAGRAM



DWG. NO.
12-LEAD 3mm P=0.5 mm
M-QFN CAVITY PACKAGE

SCALE	SIZE	DWG. NO.	REV
30:1	A	451260 M-QFN12W.5	A

RECOMMENDED PC BOARD LAYOUT DIMENSIONS IN MM



Notes: (Unless Otherwise Specified).

- 1) DIMENSIONS ARE PRESENTED ONLY AS A GUIDELINE. DESIGNERS SHOULD USE THEIR OWN KNOWLEDGE BASE WHEN DESIGNING THE PCB.
- 2) SURROUND EACH SIDE OF I/O PERIMETER PADS WITH 0.060~0.075 mm (NSMD) SOLDER MASK OPENING (2.4~3.0mils). OPTIONALLY OK TO USE RECTANGLE (NSMD) MASK OPENING AROUND I/O PADS.
- 3) ROUNDED PCB LAND PADS REDUCE SOLDER BRIDGING. PAD CHAMFER ANGLE MAY VARY
- 4) PCB LANDS SHOULD BE 0.2mm LONGER THAN THE PACKAGE I/O PADS.
- 5) THE WIDTH OF PERIMETER PCB PADS SHOULD MATCH (1:1) THE SAME WIDTH AS THE PACKAGE PADS.
- 6) REFER TO INDUSTRY REFERENCES SUCH AS IPC-SM-782 FOR PCB LAND PATTERN DESIGN.
- 7) THERMAL GROUND PADS MAY BE CHANGED TO SUITE REQUIREMENTS OF THE DESIGNER.
 - A. MAKE COPPER THERMAL PAD AS LARGE AS POSSIBLE.
 - B. DRILL MULTIPLE THERMAL VIAS 0.25~0.33mm DIAMETER USING 0.8~1.2mm PITCH GRID.
 - C. PLATE THERMAL VIA BARRELS WITH 1-OUNCE COPPER (18µm).
 - D. TENT (COVER) THERMAL VIAS WITH SOLDER MASK 0.1mm LARGER THEN THE VIA DIAMETER.
- 8) STENCIL DESIGN MAY BE CHANGED TO SUITE REQUIREMENTS OF THE DESIGNER.
 - A. LASER CUT STENCIL 0.125mm (5mil) THICK. APERTURE SIZE-TO-LAND RATIO OF 1:1.
 - B. THE SOLDER PASTE OPENING IN THE THERMAL PAD AREA SHOULD BE A MATRIX ARRAY OF SMALLER APERTURES INSTEAD OF ONE LARGE APERTURE TO CONTROL PASTE AMOUNTS.
 - C. APPLY 50% TO 80% SOLDER PASTE COVERAGE IN THE THERMAL PAD AREA.



DWG. NO.
12-LEAD 3mm P=0.5 mm
M-QFN CAVITY PACKAGE

SCALE 30:1	SIZE A	DWG. NO. 451260 M-QFN12W.5	REV A
----------------------	------------------	--	-----------------