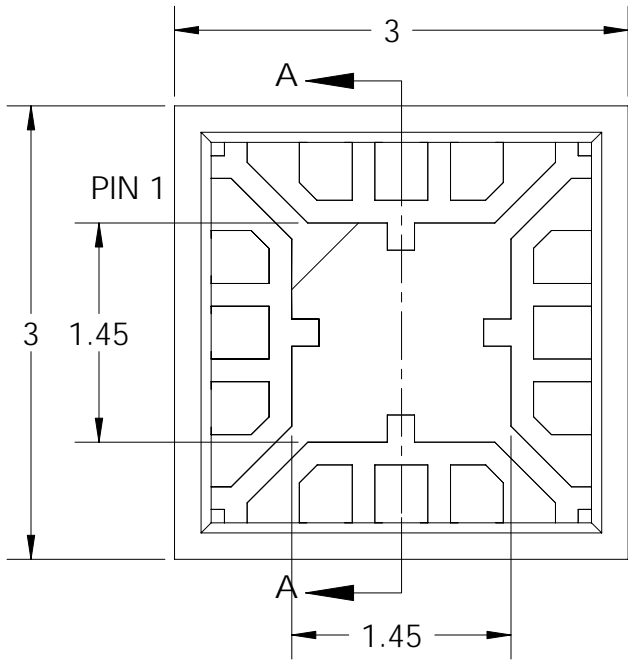
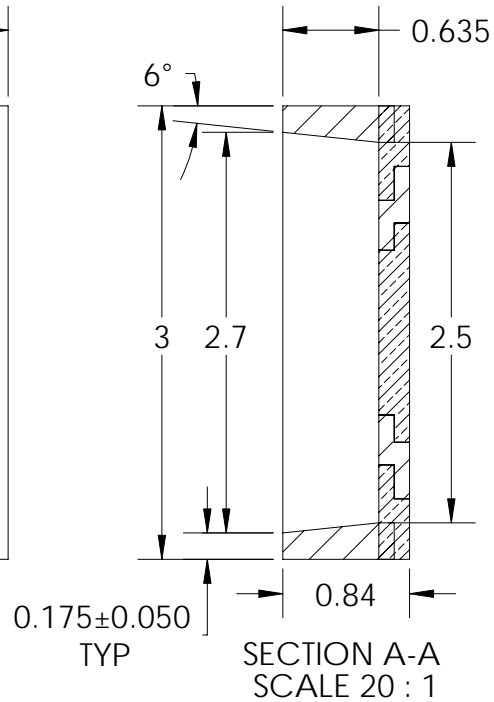


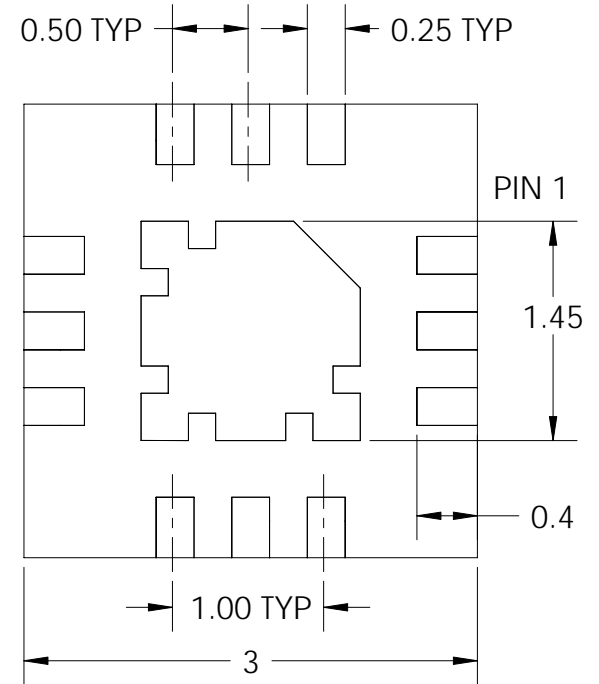
TOP VIEW



SIDE VIEW



BOTTOM VIEW



- Notes: (Unless Otherwise Specified)
- 1) BODY; PLASTIC, SEMICONDUCTOR GRADE
 - 2) LEAD FRAME: COPPER, C-194F/H
 - 3) LEAD FRAME PLATING: Ni, Pd, Au
 - 4) FRAME THICKNESS: 0.203mm
 - 5) DIE PAD: 1.45 X 1.45mm
 - 6) JEDEC OUTLINE: MO-220
 - 7) DIMENSIONS: MM

PLATING THICKNESS		
Ni	NICKEL	0.50-2.00 μm
Pd	PALLADIUM	0.02-0.15 μm
Au	GOLD	0.003-0.015 μm

TOLERANCE UNLESS NOTED		APPROVALS	DATE
X.X	± 0.05	DRAWN TST	8/21/10
X.XX	± 0.01		
X.XXX	± 0.005	CHECKED	
X.XXXX	± 0.0005	ENG	
ALL DIMENSIONS IN <input type="checkbox"/> INCHES <input type="checkbox"/> MILLIMETERS		MFG	
		Q.A.	
THIRD ANGLE PROJECTION 		CUST	
		REVISED	

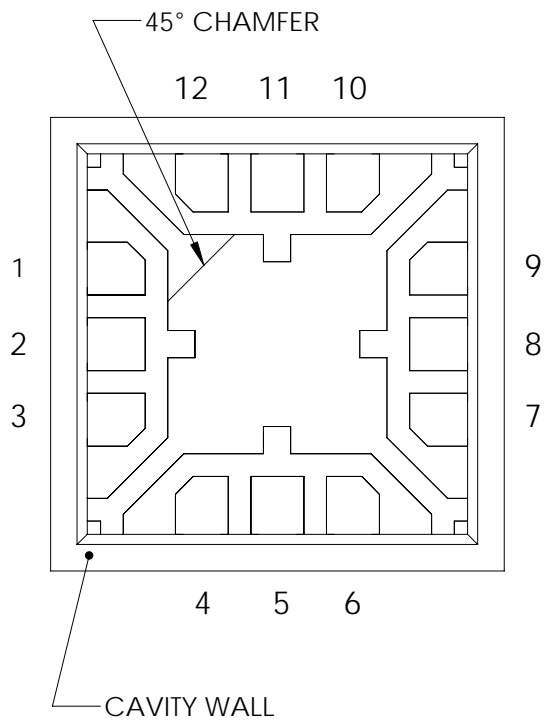


TITLE:
12-LEAD 3mm P=0.5 mm
M-QFN CAVITY PACKAGE

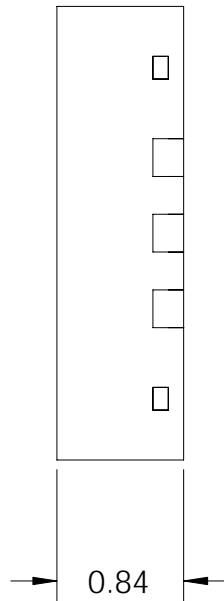
SCALE 20:1	SIZE A	DWG. NO. 451250 M-QFN12W.5	REV A
----------------------	------------------	--	-----------------

DO NOT SCALE DRAWING SHEET 1 OF 4

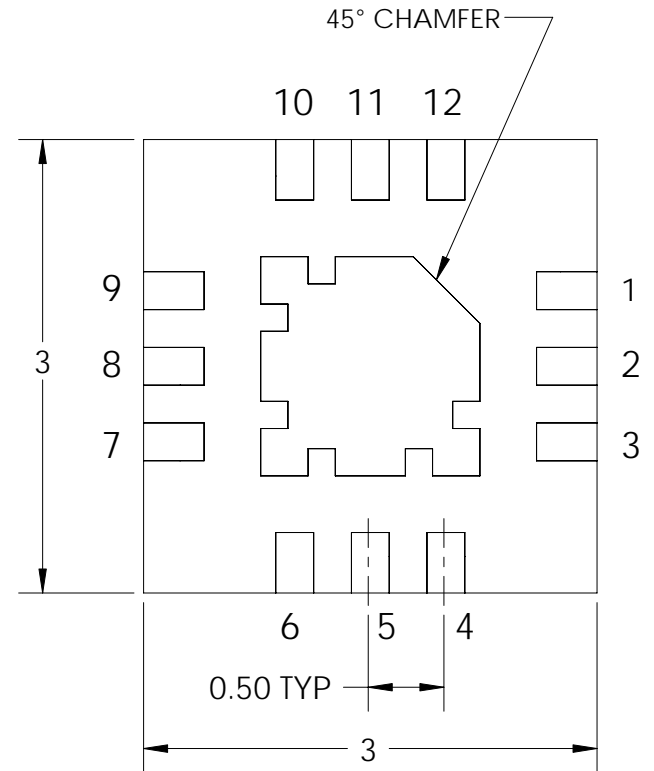
TOP VIEW



SIDE VIEW
(BEFORE LID ATTACH)



BOTTOM VIEW



TITLE:

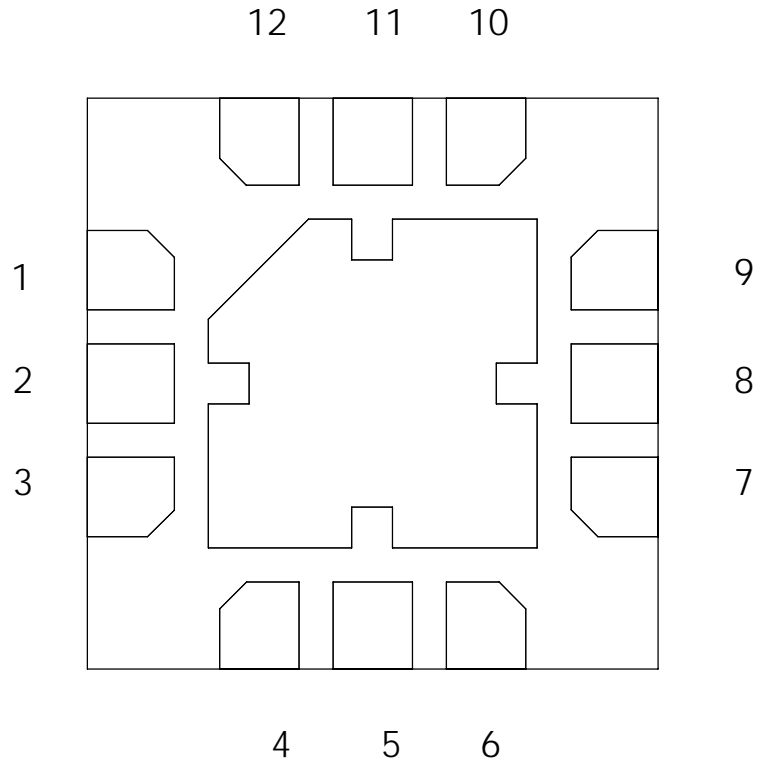
12-LEAD 3mm P=0.5 mm
M-QFN CAVITY PACKAGE

SCALE	SIZE	DWG. NO.	REV
12:1	A	451250 M-QFN12W.5	A

DO NOT SCALE DRAWING

SHEET 2 OF 4

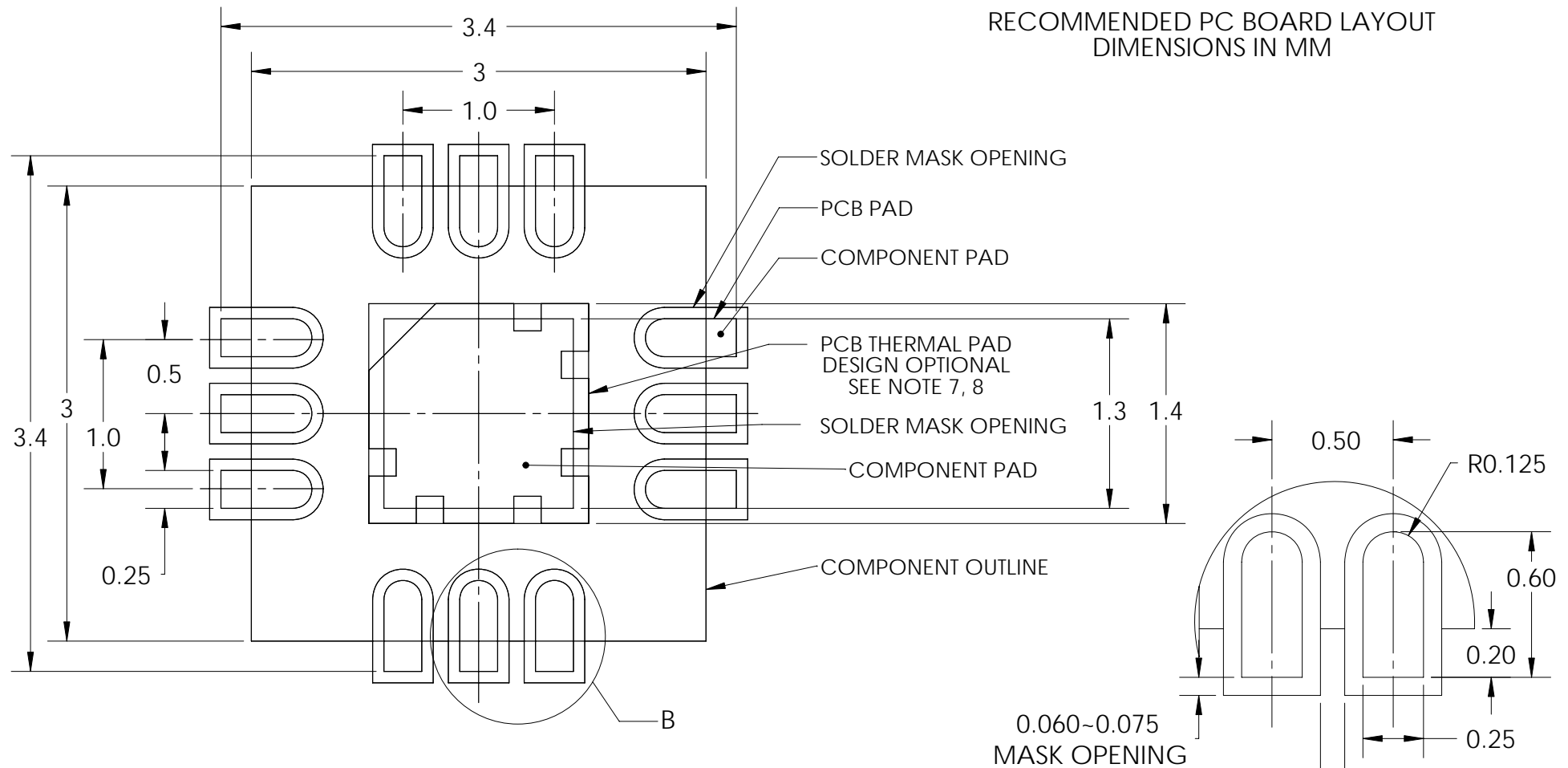
BOND DIAGRAM



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

SCALE	SIZE	DWG. NO.	REV
30:1	A	451250 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. 451250 M-QFN12W.5	REV A
----------------------	------------------	--	-----------------