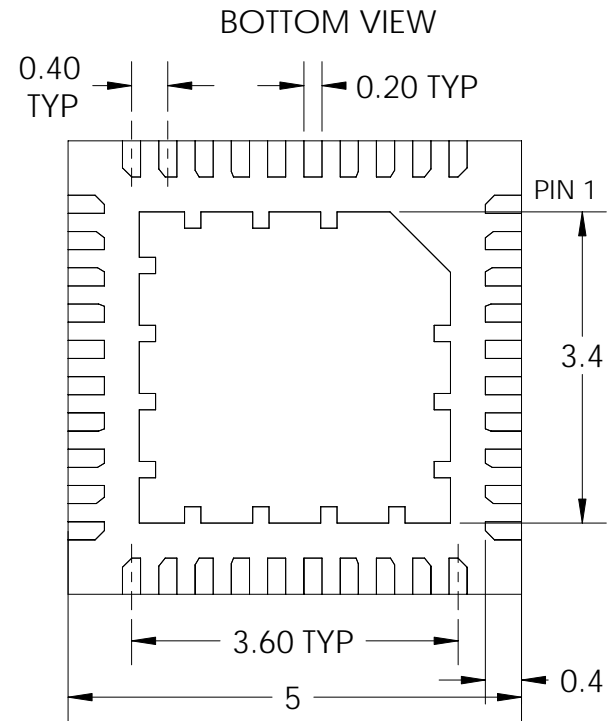

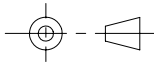


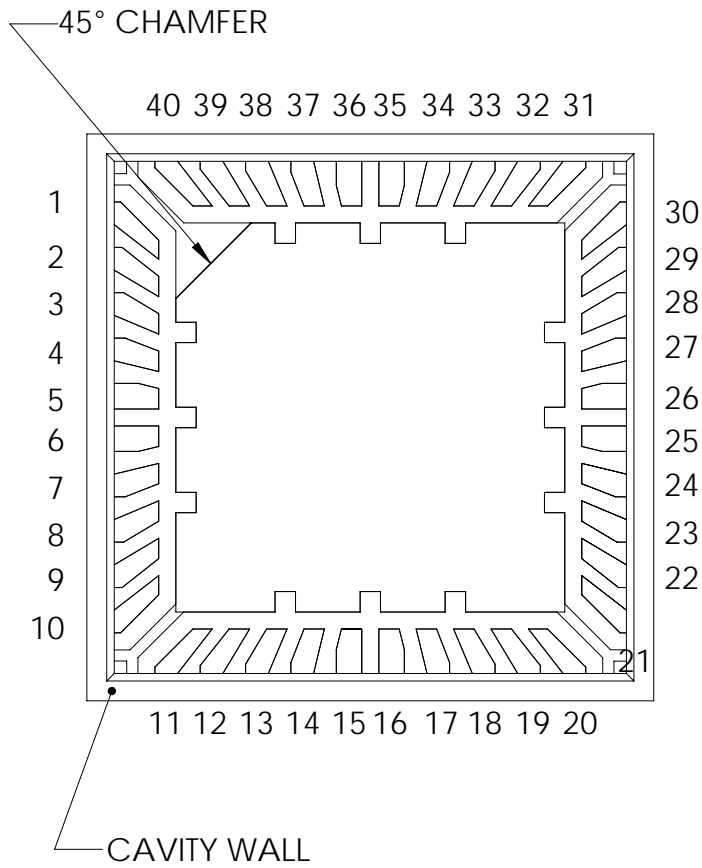
SECTION A-A  
SCALE 12 : 1



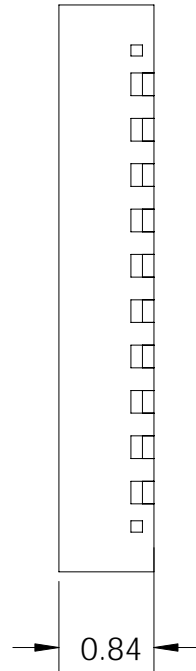
- Notes: (Unless Otherwise Specified)
- 1) BODY; PLASTIC, SEMICONDUCTOR GRADE
  - 2) LEAD FRAME: COPPER, C-194F/H
  - 3) LEAD FRAME PLATING: Ni, Pd, Au
  - 4) FRAME THICKNESS: 0.203mm
  - 5) DIE PAD: 3.4 X 3.4mm
  - 6) JEDEC OUTLINE: MO-220
  - 7) DIMENSIONS: MM

TOLERANCE UNLESS NOTED		APPROVALS	DATE	 TITLE: 40-LEAD 5mm P=0.4 mm M-QFN CAVITY PACKAGE			
X.X	± 0.05	DRAWN MH	8/21/10				
X.XX	± 0.01	CHECKED					
X.XXX	± 0.005	ENG					
X.XXXX	± 0.0005	MFG					
ALL DIMENSIONS IN <input type="checkbox"/> INCHES <input type="checkbox"/> MILLIMETERS		Q.A.		SCALE	SIZE	DWG. NO.	REV
THIRD ANGLE PROJECTION		CUST		12:1	A	444050 M-QFN40W.4	A
		REVISED		DO NOT SCALE DRAWING		SHEET 1 OF 4	

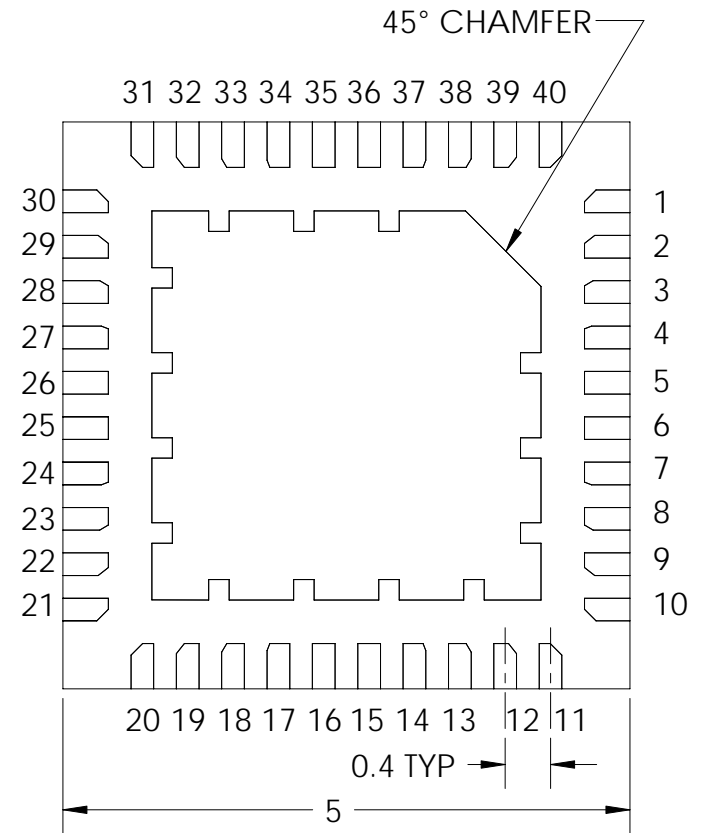
TOP VIEW



SIDE VIEW  
(BEFORE LID ATTACH)



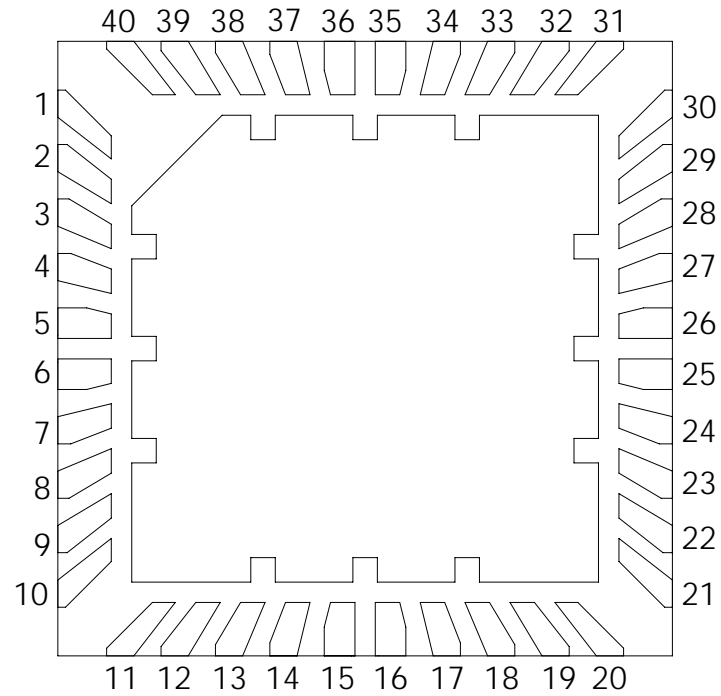
BOTTOM VIEW



TITLE:  
40-LEAD 5mm P=0.4 mm  
M-QFN CAVITY PACKAGE

SCALE <b>15:1</b>	SIZE <b>A</b>	DWG. NO. <b>444050 M-QFN40W.4</b>	REV <b>A</b>
----------------------	------------------	--	-----------------

# BONDING DIAGRAM



**Mirror**  
Semiconductor™

TITLE:

40-LEAD 5mm P=0.4 mm  
M-QFN CAVITY PACKAGE

SCALE  
**18:1**

SIZE  
**A**

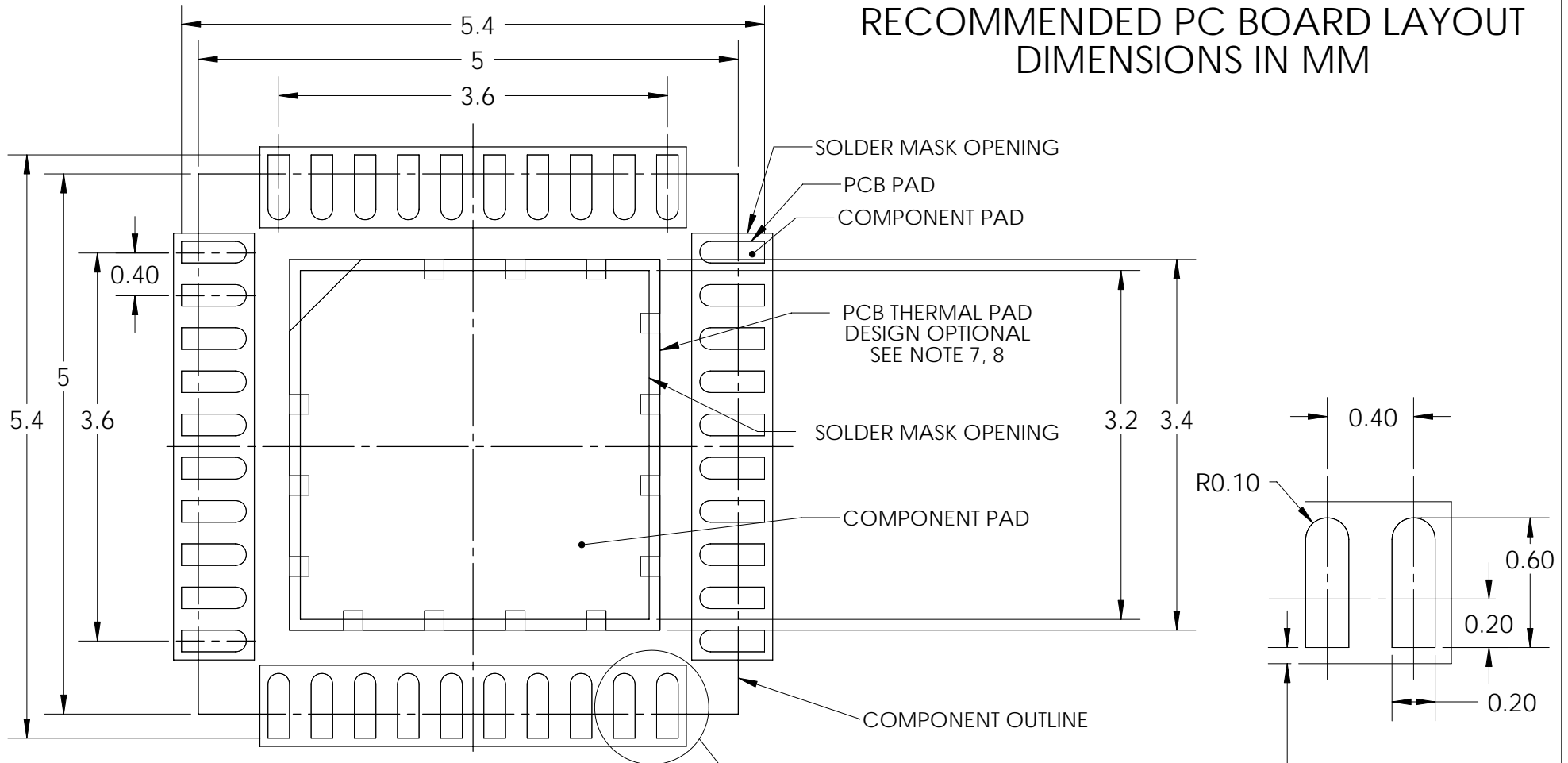
DWG. NO.  
**444050**  
**M-QFN40W.4**

REV  
**A**

DO NOT SCALE DRAWING

SHEET 3 OF 4

# 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 (18 $\mu$ m).
  - 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.

0.060~0.075  
MASK OPENING

DETAIL B  
SCALE 36 : 1



TITLE:

40-LEAD 5mm P=0.4 mm  
M-QFN CAVITY PACKAGE

SCALE	SIZE	DWG. NO.	REV
18:1	A	444050 M-QFN40W.4	A

DO NOT SCALE DRAWING

SHEET 4 OF 4