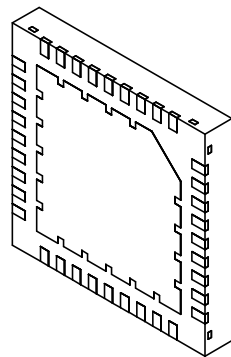

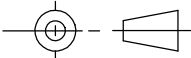


ISOMETRIC VIEW OF CAVITY SIDE

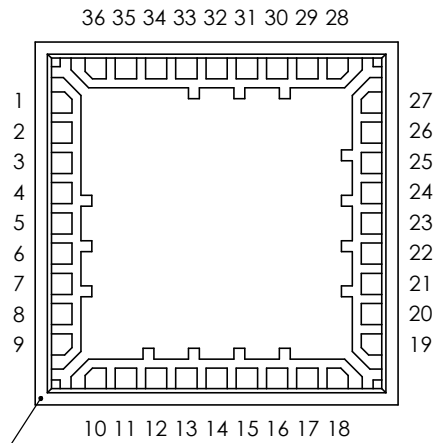


ISOMETRIC VIEW OF PCB SIDE

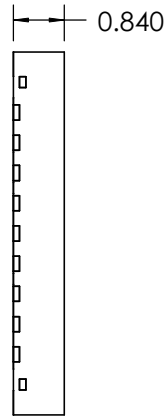
- Notes: (Unless Otherwise Specified)
- 1) BODY; PLASTIC, SEMICONDUCTOR GRADE
  - 2) LEAD FRAME: COPPER, C-194F/H
  - 3) LEAD FRAME PLATING: Ni, Pd, Au
  - 4) FRAME THICKNESS: 0.203mm
  - 5) DIE PAD: 6.39 X 6.39mm
  - 6) JEDEC OUTLINE: MO-220

TOLERANCES UNLESS NOTED		APPROVALS		DATE	 www.MirrorSemi.com			
X.X	± 0.05	DRAWN	EDK	07/17/2012				
X.XX	± 0.01	CHECKED			TITLE: <b>36-LEAD 6mm P=0.5mm QFN CAVITY PACKAGE</b>			
X.XXX	± 0.005	ENG APPR.						
X.XXXX	± 0.0005	MFG APPR.						
ALL DIMENSIONS IN		Q.A.						
□ INCHES    ✕ MILLIMETERS		CUST.			SCALE	SIZE	DWG. NO.	REV
THIRD ANGLE PROJECTION		REVISED			<b>8:1</b>	<b>A</b>	<b>453670</b> <b>M-QFN36W.5</b>	<b>A</b>
					DO NOT SCALE DRAWING		SHEET 1 OF 4	

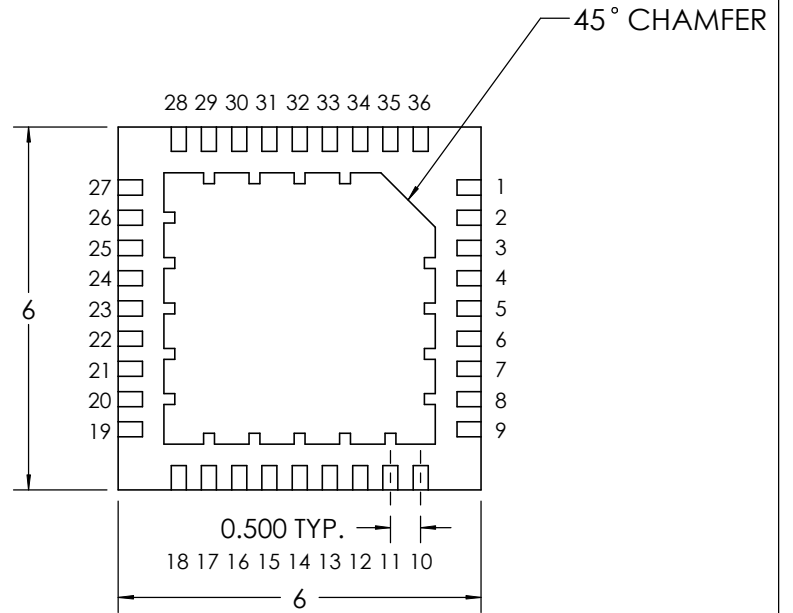
TOP VIEW



SIDE VIEW  
(BEFORE LID ATTACH)



BOTTOM VIEW



**Mirror**  
Semiconductor™

www.MirrorSemi.com

TITLE:

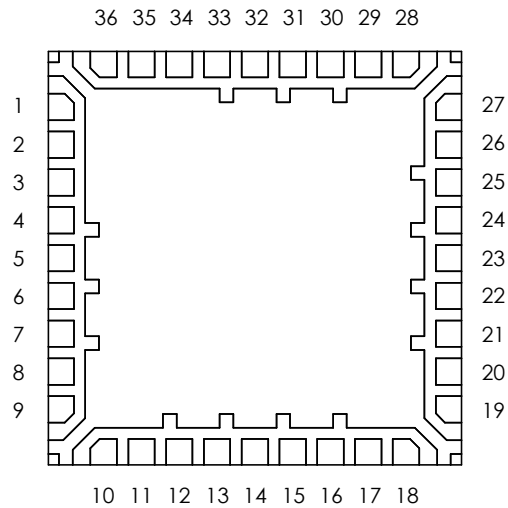
36-LEAD 6mm P=0.5mm  
QFN CAVITY PACKAGE  
LEAD NUMBERING

SCALE	SIZE	DWG. NO.	REV
8:1	A	453670 M-QFN36W.5	A

DO NOT SCALE DRAWING

SHEET 2 OF 4

# BOND DIAGRAM



**Mirror**  
Semiconductor™

www.MirrorSemi.com

TITLE:

36-LEAD 6mm P=0.5mm  
QFN CAVITY PACKAGE  
BOND DIAGRAM

SCALE

10:1

SIZE

**A**

DWG. NO.

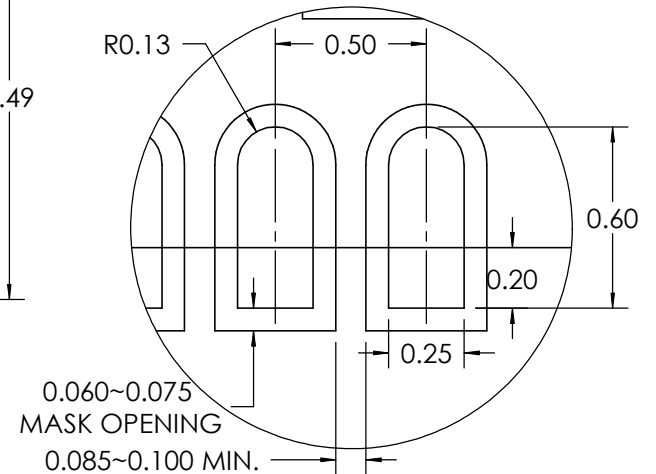
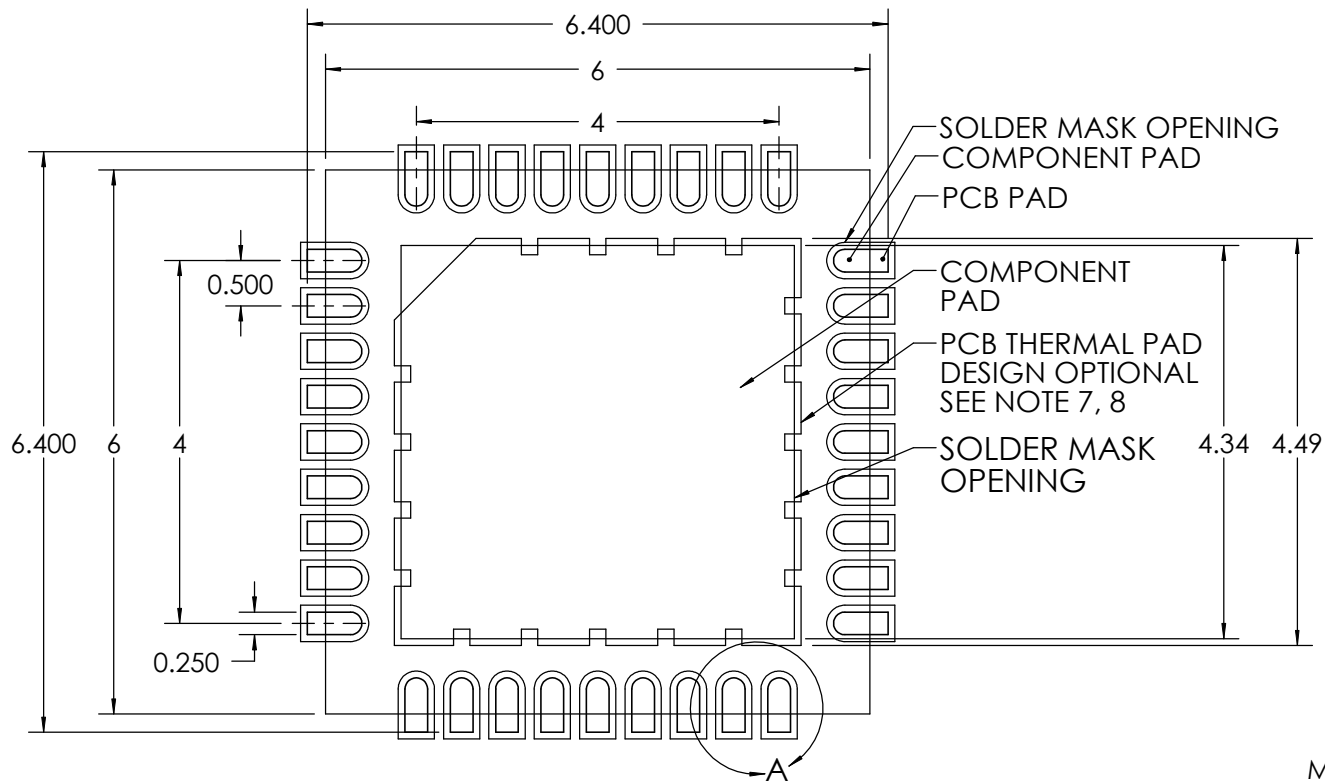
**453670**  
**M-QFN36W.5**

REV

**A**

DO NOT SCALE DRAWING

SHEET 3 OF 4



DETAIL A  
SCALE 40 : 1

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 (2.4~3.0mils) NSMD SOLDER MASK OPENING. 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 WIDTH OF 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 THAN 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 PAD AREA.

**Mirror**  
Semiconductor™

www.MirrorSemi.com

TITLE: 36-LEAD 6mm P=0.5mm  
QFN CAVITY PACKAGE  
RECOMMENDED PCB  
LAYOUT

SCALE	SIZE	DWG. NO.	REV
12:1	A	453670 M-QFN36W.5	A

DO NOT SCALE DRAWING SHEET 4 OF 4