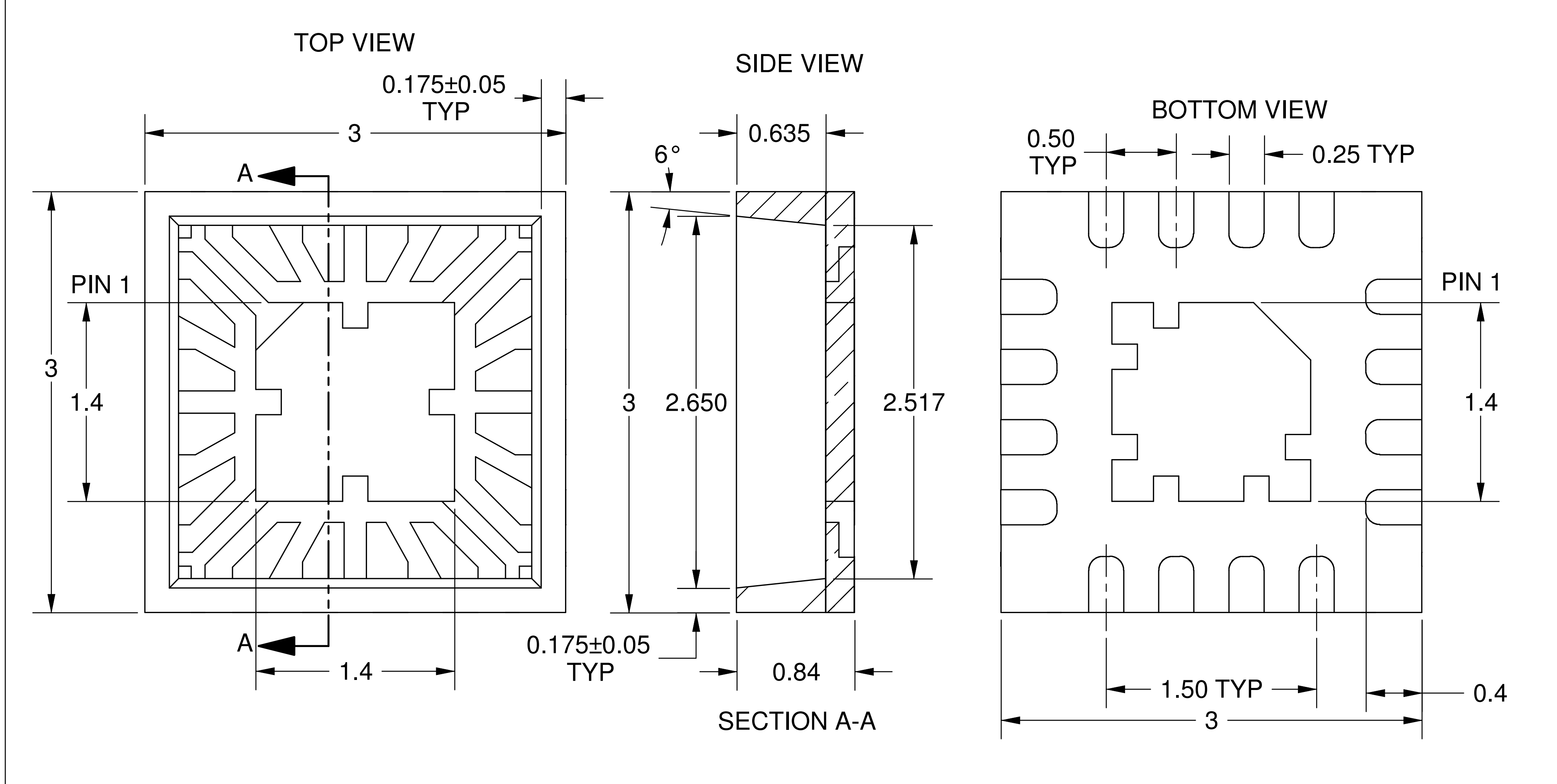


:

Notes: (Unless Otherwise Specified).
1) BODY: PLASTIC, SEMICONDUCTOR GRADE.
2) LEAD FRAME: COPPER, C-194 F/H.
3) LEAD FRAME PLATING: Ni, Pd, Au.
4) FRAME THICKNESS: 0.203MM.
5) DIE PAD: 1.4 X 1.4MM.
6) JEDEC OUTLINE: MO-220.
7) DIMENSIONS: MM.

TOLERANCE UNLESS NOTED		APPROVALS	DATE
X.X	+/- 0.05		
X.XX	+/- 0.01	DRAWN MH	6/14/2010
X.XXX	+/- 0.005		
ANGLES	+/- 0.5°	ENG	
ALL DIMENSIONS IN ☒ INCHES ☐ MILLIMETERS		MFG	
THIRD ANGLE PROJECTION		QA	
		CUST	
		REVISED	

Mirror Semiconductor™

TITLE 16-LEAD 3mm P=0.5 mm M-QFN CAVITY PACKAGE

SCALE	SIZE	DRAWING NO.	REV
24:1	A	451650	A

DO NOT SCALE DRAWING

PIN LOCATIONS

TOP VIEW

45° CHAMFER

16 15 14 13

1 2 3 4

12 11 10 9

5 6 7 8

CAVITY WALL

SIDE VIEW (BEFORE LID ATTACH)

0.84

BOTTOM VIEW

45° CHAMFER

13 14 15 16

12 11 10 9

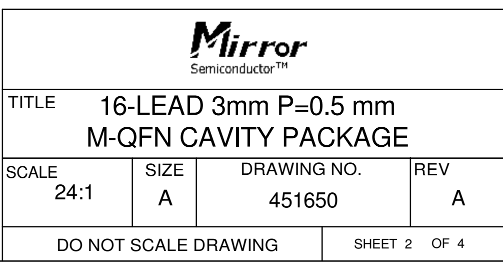
1 2 3 4

8 7 6 5

3

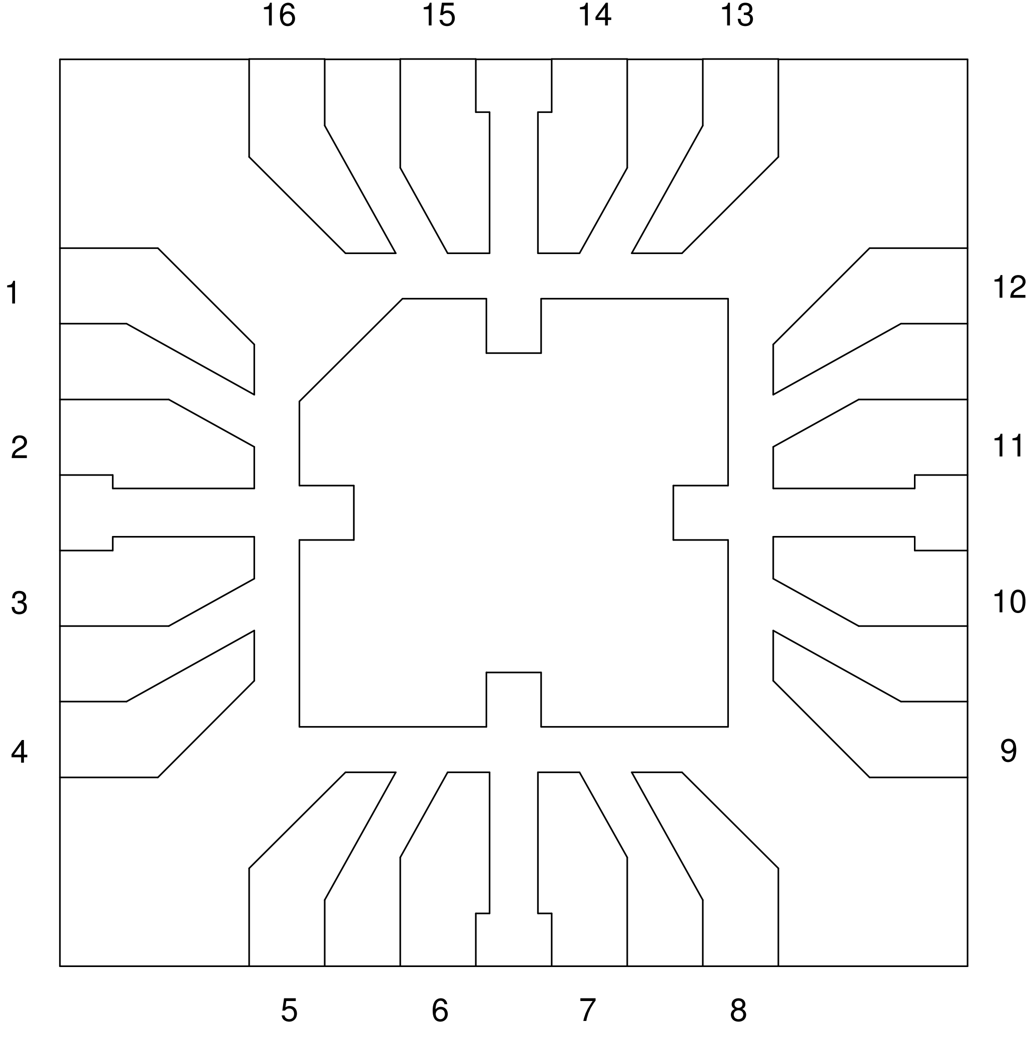
0.50 TYP

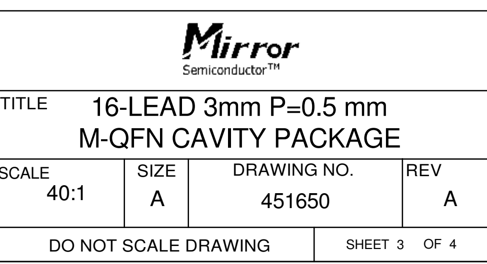
3



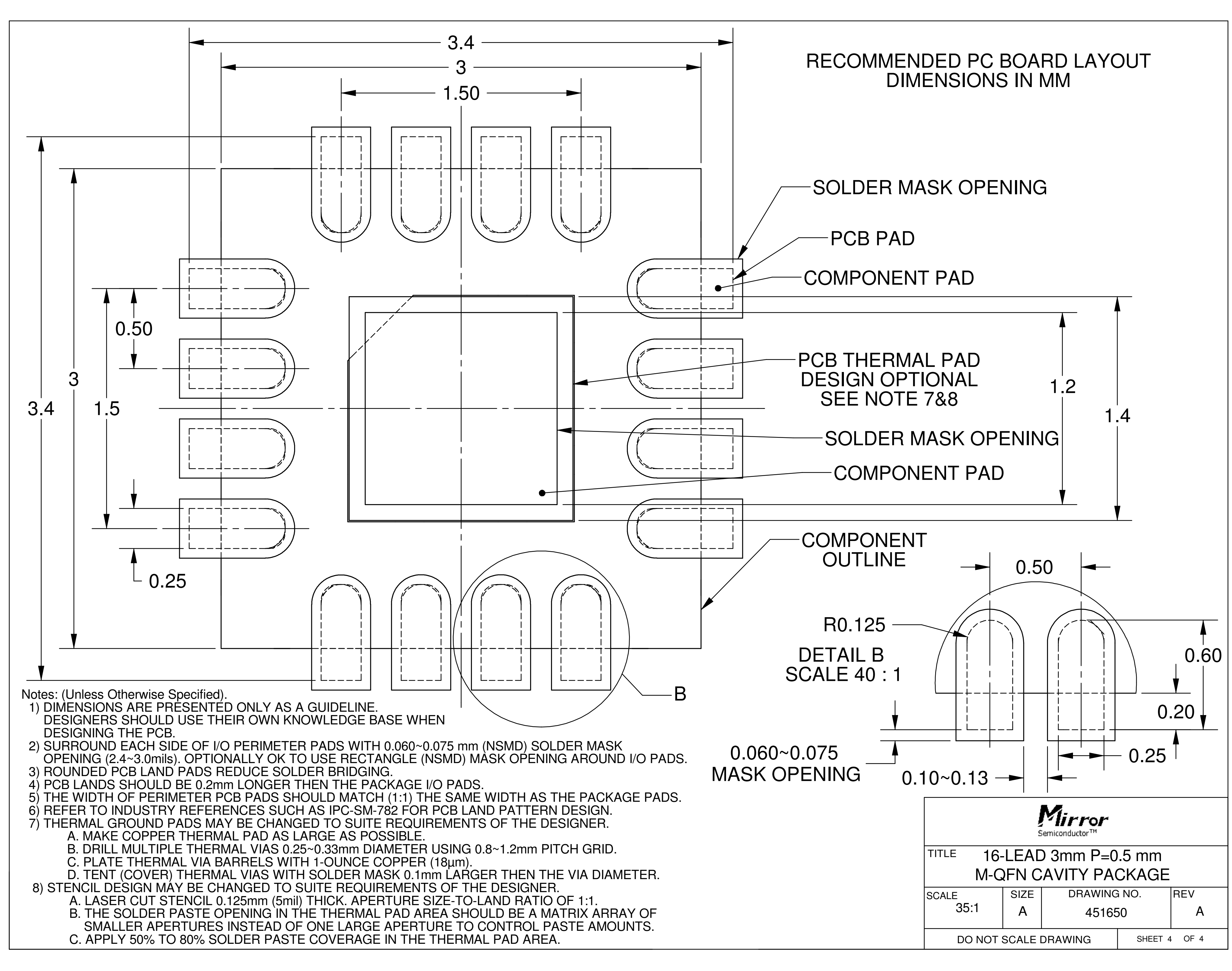
TITLE	16-LEAD 3mm P=0.5 mm M-QFN CAVITY PACKAGE		
SCALE 24:1	SIZE A	DRAWING NO. 451650	REV A
DO NOT SCALE DRAWING		SHEET 2 OF 4	

BONDING DIAGRAM





TITLE	16-LEAD 3mm P=0.5 mm M-QFN CAVITY PACKAGE		
SCALE 40:1	SIZE A	DRAWING NO. 451650	REV A
DO NOT SCALE DRAWING		SHEET 3 OF 4	



RECOMMENDED PC BOARD LAYOUT
DIMENSIONS IN MM
3.4
3
1.50
SOLDER MASK OPENING
PCB PAD
COMPONENT PAD
0.50
3
3.4
1.5
PCB THERMAL PAD
DESIGN OPTIONAL
SEE NOTE 7&8
1.2
1.4
SOLDER MASK OPENING
COMPONENT PAD
0.25
COMPONENT
OUTLINE
0.50
R0.125
DETAIL B
SCALE 40 : 1
0.60
B
0.20
0.060~0.075
MASK OPENING
0.25
0.10~0.13
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.
4) PCB LANDS SHOULD BE 0.2mm LONGER THEN 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.
Mirror
Semiconductor™
TITLE
16-LEAD 3mm P=0.5 mm
M-QFN CAVITY PACKAGE
SCALE
35:1
SIZE
A
DRAWING NO.
451650
REV
A
DO NOT SCALE DRAWING
SHEET 4 OF 4