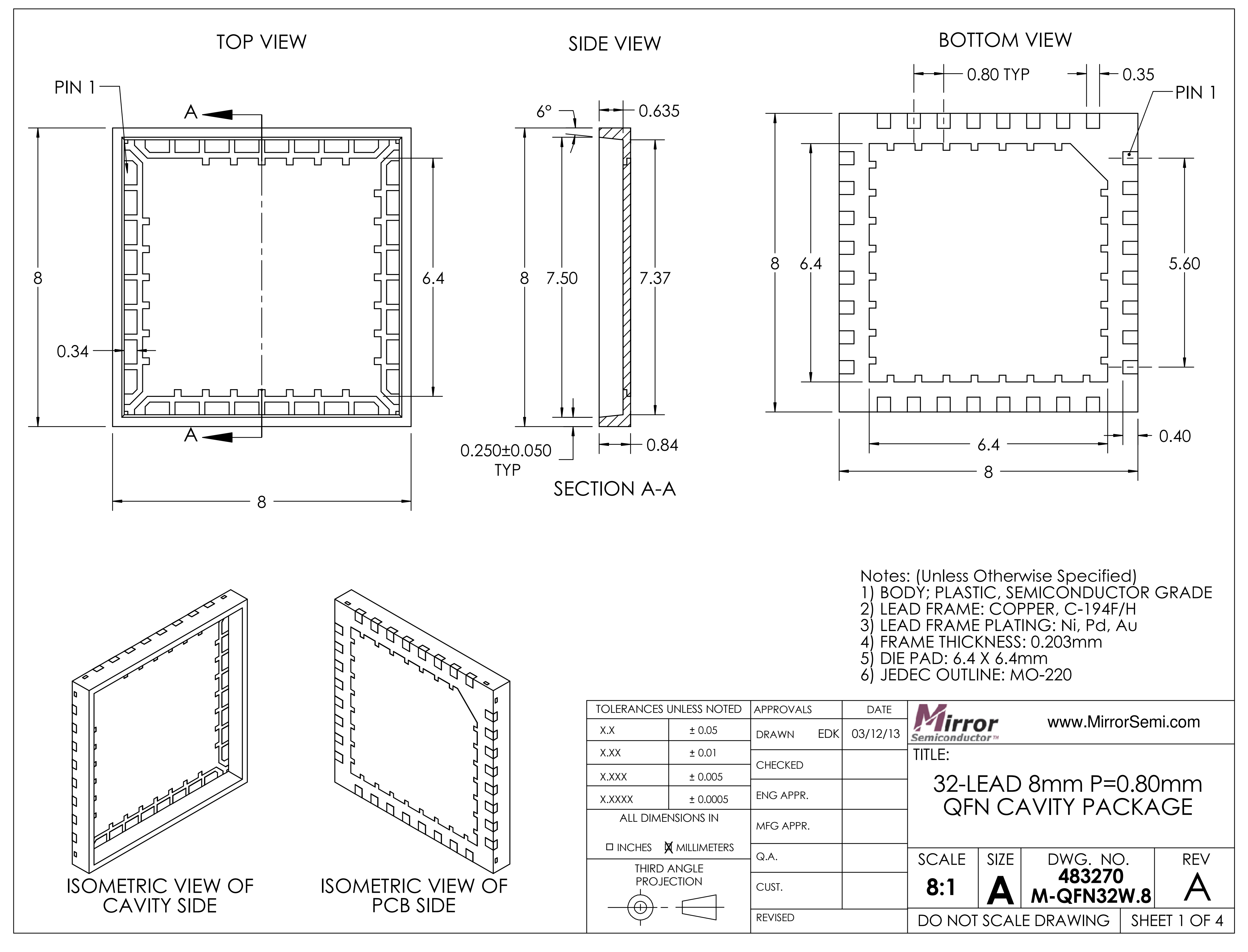


TOP VIEW
PIN 1
A
8
6.4
0.34
A
8
SIDE VIEW
6°
0.635
8
7.50
7.37
0.250±0.050
TYP
0.84
SECTION A-A
BOTTOM VIEW
0.80 TYP
0.35
PIN 1
8
6.4
5.60
6.4
0.40
8
ISOMETRIC VIEW OF
CAVITY SIDE
ISOMETRIC VIEW OF
PCB SIDE
Notes: (Unless Otherwise Specified)
1) BODY; PLASTIC, SEMICONDUCTOR GRADE
2) LEAD FRAME: COPPER, C-194F/H
3) LEAD FRAME PLATING: Ni, Pd, Au
4) FRAME THICKNESS: 0.203mm
5) DIE PAD: 6.4 X 6.4mm
6) JEDEC OUTLINE: MO-220
TOLERANCES UNLESS NOTED
X.X ± 0.05
X.XX ± 0.01
X.XXX ± 0.005
X.XXXX ± 0.0005
ALL DIMENSIONS IN
INCHES MILLIMETERS
THIRD ANGLE
PROJECTION
APPROVALS
DATE
DRAWN EDK 03/12/13
CHECKED
ENG APPR.
MFG APPR.
Q.A.
CUST.
REVISED
Mirror
Semiconductor™
www.MirrorSemi.com
TITLE:
32-LEAD 8mm P=0.80mm
QFN CAVITY PACKAGE
SCALE
8:1
SIZE
A
DWG. NO.
483270
M-QFN32W.8
REV
A
DO NOT SCALE DRAWING
SHEET 1 OF 4

TOP VIEW

32 31 30 29 28 27 26 25

1 2 3 4 5 6 7 8

24 23 22 21 20 19 18 17

9 10 11 12 13 14 15 16

CAVITY WALL

SIDE VIEW
(BEFORE LID ATTACH)

BOTTOM VIEW

45° CHAMFER

25 26 27 28 29 30 31 32

24 23 22 21 20 19 18 17

1 2 3 4 5 6 7 8

8

0.80

16 15 14 13 12 11 10 9

8

Mirror Semiconductor™ www.MirrorSemi.com

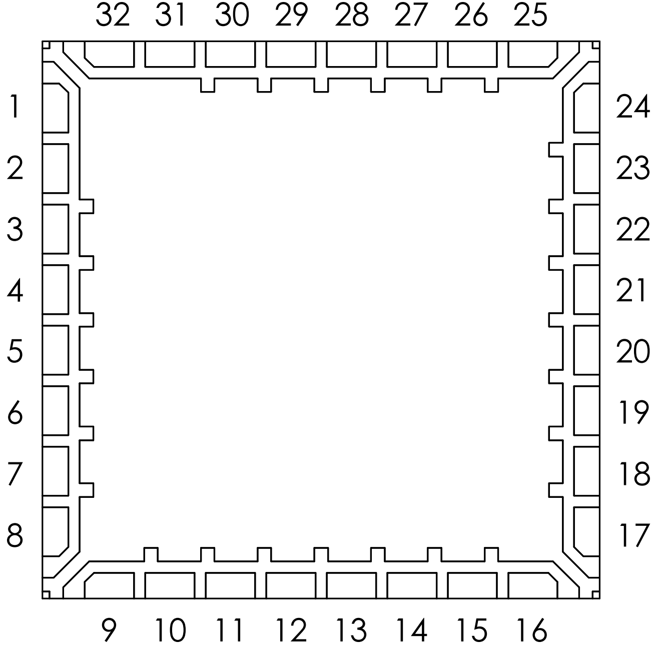
TITLE:

32-LEAD 8mm P=0.80mm
QFN CAVITY PACKAGE
LEAD NUMBERING

SCALE	SIZE	DWG. NO.	REV
8:1	**A**	**483270 M-QFN32W.8**	A

DO NOT SCALE DRAWING	SHEET 2 OF 4

BOND DIAGRAM



8.40
8
5.60
SOLDER MASK OPENING
COMPONENT PAD
PCB PAD
COMPONENT PAD
PCB THERMAL PAD DESIGN OPTIONAL SEE NOTE 7, 8
SOLDER MASK OPENING
0.80
8.40 8 5.60
6.24 6.4
0.35
A

0.80
R0.18
0.60
0.20
0.35
0.060~0.075 MASK OPENING
0.185~0.300 MIN.

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 (18μm).
   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 APERATURES INSTEAD OF ONE LARGE APERATURE TO CONTROL PASTE AMOUNTS.
   C. APPLY 50% TO 80% SOLDER PASTE COVERAGE IN THE PAD AREA.

www.MirrorSemi.com

TITLE: 32-LEAD 8mm P=0.80mm QFN CAVITY PACKAGE RECOMMENDED PCB LAYOUT

SCALE	SIZE	DWG. NO.	REV
10:1	A	483270 M-QFN32W.8	A

DO NOT SCALE DRAWING