TOP VIEW

0.175±0.05
TYP

4

A

PIN 1

4

2.4

A

2.4

SIDE VIEW

6°

0.635

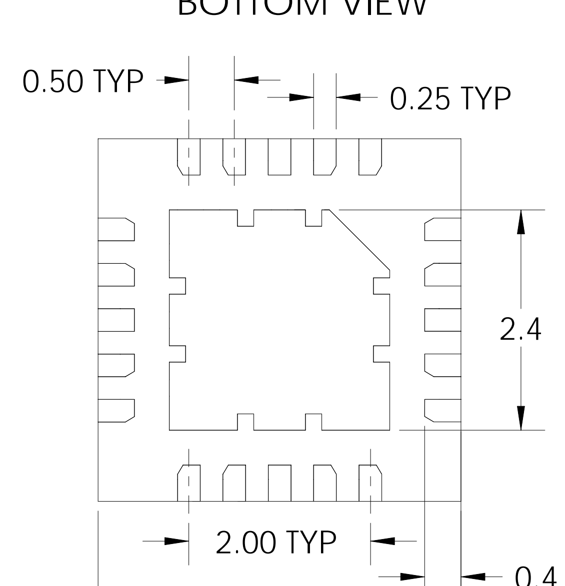
4

3.7

3.5

0.175±0.05
TYP

0.84



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: 2.4 X 2.4mm
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	
X.X	± 0.05
X.XX	± 0.01
X.XXX	± 0.005
X.XXXX	± 0.0005
ALL DIMENSIONS IN ☐INCHES ☒MILLIMTERS	
THIRD ANGLE PROJECTION	

APPROVALS	DATE
DRAWN	8/21/10
CHECKED	
ENG	
MFG	
Q.A.	
CUST	
REVISED	

Mirror Semiconductor™

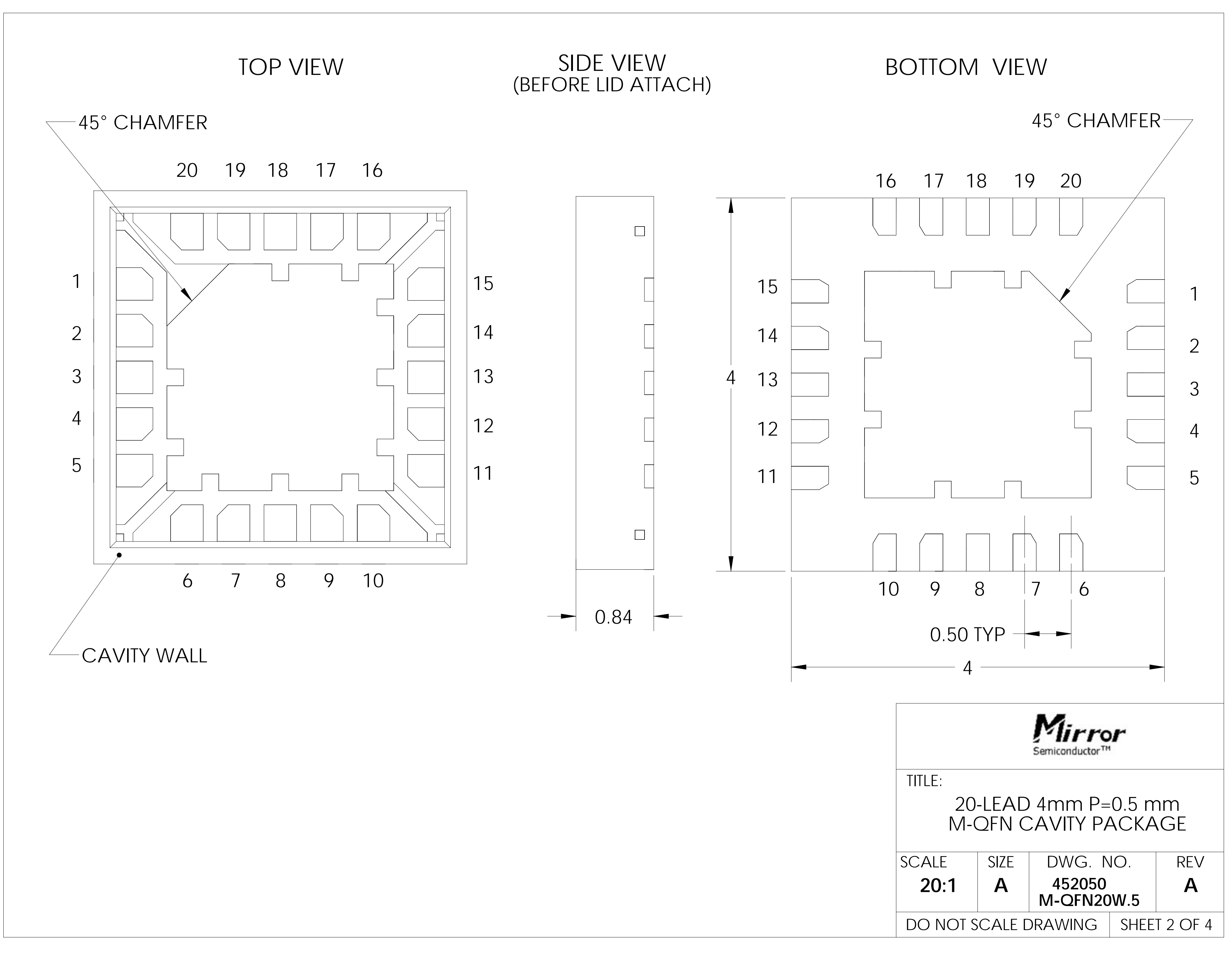
TITLE:

20-LEAD 4mm P=0.5 mm
M-QFN CAVITY PACKAGE

SCALE	SIZE	DWG. NO.	REV
12:1	A	452050 M-QFN20W.5	A

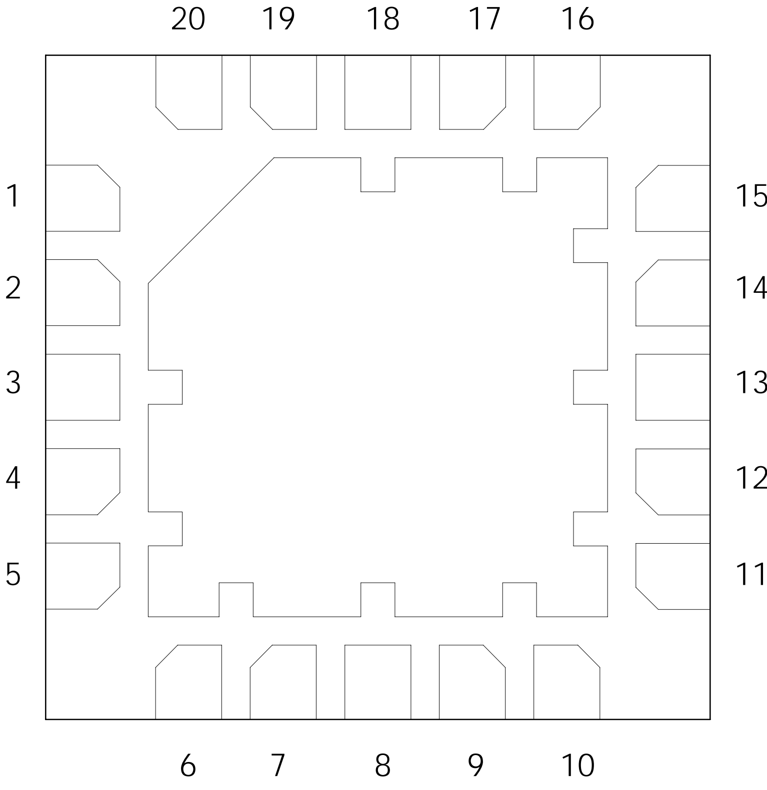
DO NOT SCALE DRAWING

5 4 3 2 1



TOP VIEW
SIDE VIEW
(BEFORE LID ATTACH)
BOTTOM VIEW
45° CHAMFER
20 19 18 17 16
1 2 3 4 5
15 14 13 12 11
6 7 8 9 10
CAVITY WALL
0.84
45° CHAMFER
16 17 18 19 20
15 14 13 12 11
1 2 3 4 5
4
10 9 8 7 6
0.50 TYP
4
Mirror
Semiconductor™
TITLE:
20-LEAD 4mm P=0.5 mm
M-QFN CAVITY PACKAGE
SCALE
20:1
SIZE
A
DWG. NO.
452050
M-QFN20W.5
REV
A
DO NOT SCALE DRAWING
SHEET 2 OF 4

BONDING DIAGRAM



TITLE:

20-LEAD 4mm P=0.5 mm
M-QFN CAVITY PACKAGE

SCALE	SIZE	DWG. NO.	REV
18:1	**A**	**452050 M-QFN20W.5**	**A**

DO NOT SCALE DRAWING

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.

TITLE:
20-LEAD 4mm P=0.5 mm
M-QFN CAVITY PACKAGE

SCALE	SIZE	DWG. NO.	REV
18:1	A	452050 M-QFN20W.5	A

DO NOT SCALE DRAWING	SHEET 4 OF 4