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: 4.4 X 4.4mm
6) JEDEC OUTLINE: MO-220
7) DIMENSIONS: MM

TOLERANCE UNLESS NOTED

<table>
<thead>
<tr>
<th>TOLERANCE</th>
<th>APPROVALS</th>
<th>DATE</th>
</tr>
</thead>
<tbody>
<tr>
<td>X.X</td>
<td>± 0.05</td>
<td></td>
</tr>
<tr>
<td>X.XX</td>
<td>± 0.1</td>
<td></td>
</tr>
<tr>
<td>X.XXX</td>
<td>± 0.005</td>
<td></td>
</tr>
<tr>
<td>X.XXXXX</td>
<td>± 0.0005</td>
<td></td>
</tr>
</tbody>
</table>

ALL DIMENSIONS IN MILLIMETERS
THIRD ANGLE PROJECTION

SCALE: 10:1
SIZE: A
DWG. NO.: 444850 M-QFN48W.4
REV: A

TITLE:
48-LEAD 6mm P=0.4 mm M-QFN CAVITY PACKAGE

M-QFN Miroc
**Title:**

48-LEAD 6mm P=0.4 mm M-QFN CAVITY PACKAGE

**Scale:** 15:1

**DWG. No.:** 444850 M-QRN48W.4

**Rev.:** A

**Sheet:** 2 of 4

---

**Top View**

- 45° Chamfer
- Cavity Wall

**Side View**

(Before Lid Attach)

- 45° Chamfer

**Bottom View**

- 45° Chamfer

---

**Dimensions:**

- 6 mm
- 0.84
- 0.40 Typ
**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.**
   - **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.**
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.**