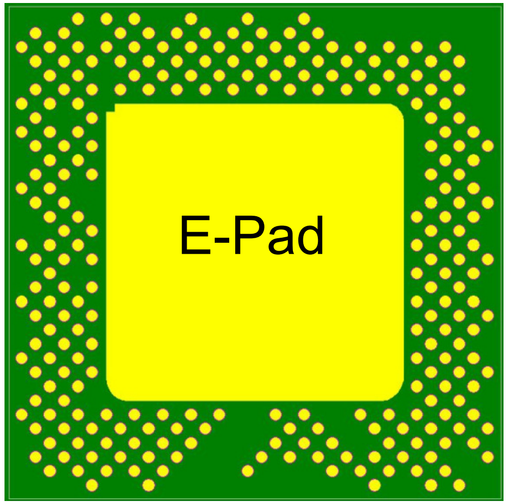
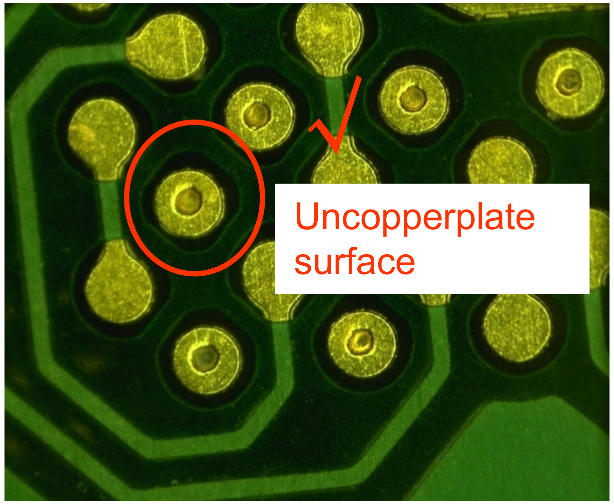
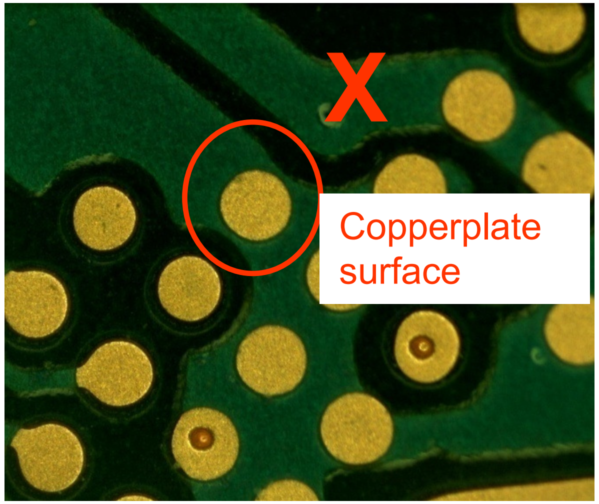
PCB Recommended Design

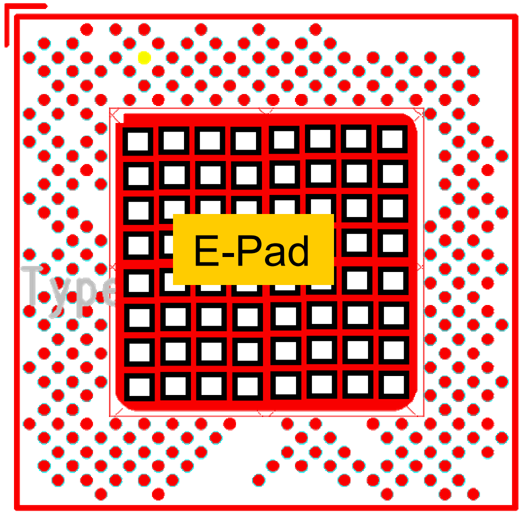






- **PCB Layout suggestion for E-pad area:**
  - IC’s E-PAD area = PCB’s E-PAD area
- **PCB Layout suggestion for Signal Pad:**
  - Pad diameter is 0.27±0.02mm.
  - Copperplate surface on pads are prohibited.
  - To ensure GND function, the trace which connects GND must use flood over design or via to inner GND plate design. And Line width not more than 0.2mm is recommended.
  - Solder-bridges must be designed between Pads.

Stencil Recommended Design



MT6253	Thickness	Opening
E-Pad	0.1mm	30%-40% Area
Signal Pad	0.1mm	0.27 ~ 0.28 Square or round shape

- **Stencil Opening suggestion for E-pad area:**
  - Stencil's E-pad opening area = 30%~40% PCB's E-pad area.
  - The E-pad edge distance between PCB and stencil is more than 0.2mm.
  - Stencil's E-pad must be divided into squares whose length within 2mm.
- **Stencil Opening suggestion for Signal Pad.**
  - Stencil thickness is 0.1mm.
  - Each diameter is 0.27~0.28mm.
  - Its figure is round or square shape.