

Precautions on printed circuit board (PCB) design

Upon the design of printed circuit board, consideration should be taken to suppress EMI level by prevention of the decrease of negative resistance.

1. Pattern length on PCB

In order to prevent degradation of the electrical characteristics by inductance and stray capacitance of wiring, it is recommended to connect crystal units with IC and capacitors by the shortest wiring length. This pattern length should be approximately within 2 cm, but the shorter the length the less EMI radiates as far as the placement of components such as IC and crystal units does not become problem. Since EMI mainly occurs from the OUT side of the inverter, the pattern length of this portion should be the shortest in crystal oscillation circuit. Also, through holes are not preferable since they become the source of EMI.

2. Influence of patterns other than crystal oscillation circuit

In the case of designing a multi-layer PCB, it is important not to design field ground or other signal patterns under oscillation circuit in the circled portion of the figure in order to prevent reduction of negative resistance, and to obtain stable starting characteristics. Especially, if other signal lines are located close to the IN side of oscillation circuit, the oscillation waveform will be modulated causing noise, and amplified at the OUT side and becomes the cause of EMI. Since crystal oscillation circuit may stop starting if the voltage of the IN terminal and the OUT terminal becomes equal under the influence of other signal lines, it is strongly advised not to design this way.

It also should be avoided to place field ground on the layer near oscillation circuit, since it will decrease negative resistance greatly.

3. Shielding with field ground

When shielding with field ground, it will be placed on the farthest layer from the component side as shown in the left figure below.

Ground pattern should not be placed in oscillation circuit in order to avoid negative resistance decrease, and other signal lines also should not be placed in this portion, since it will modulate the oscillation waveform and increase EMI by decreasing negative resistance. If the ground pattern of the oscillation circuit on the component side is placed close to signal lines, EMI level will decrease, but if is placed excessively too close, it will decrease the negative resistance, and it is advised to place it apart from the signal line of the oscillation circuit 0.5mm or more. It is not favorable to place ground pattern too close to the signal line of the oscillation circuit since it will decrease negative resistance, and besides, it is not necessary to do so since the EMI level radiated from the IN side is rather low.

