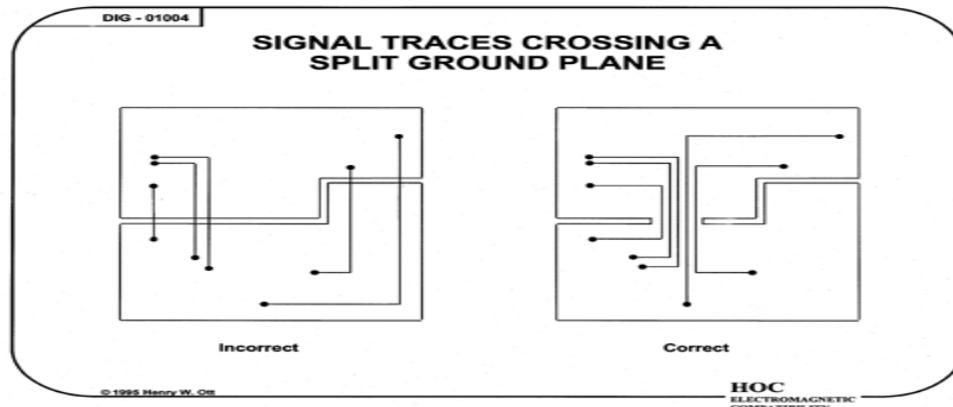


D- Know How to Use Your Power and Ground Planes:



Using a 2-layer PCB with a ground plane is also preferable to just routing ground traces. Routing a return trace adjacent to a signal trace is one strategy to minimize EMI susceptibility in multi-board designs when high speed/high frequency signals are routed between boards through a cable. Placing a ground return as close as possible to a signal line allows the return signal to closely follow the trace, which minimizes loop inductance and EMI susceptibility.

Placing a ground plane below the relevant components and signal traces also allows you to route ground returns directly from a component to the ground plane through a via. The return signal will then follow the path of least impedance back to the power supply return.