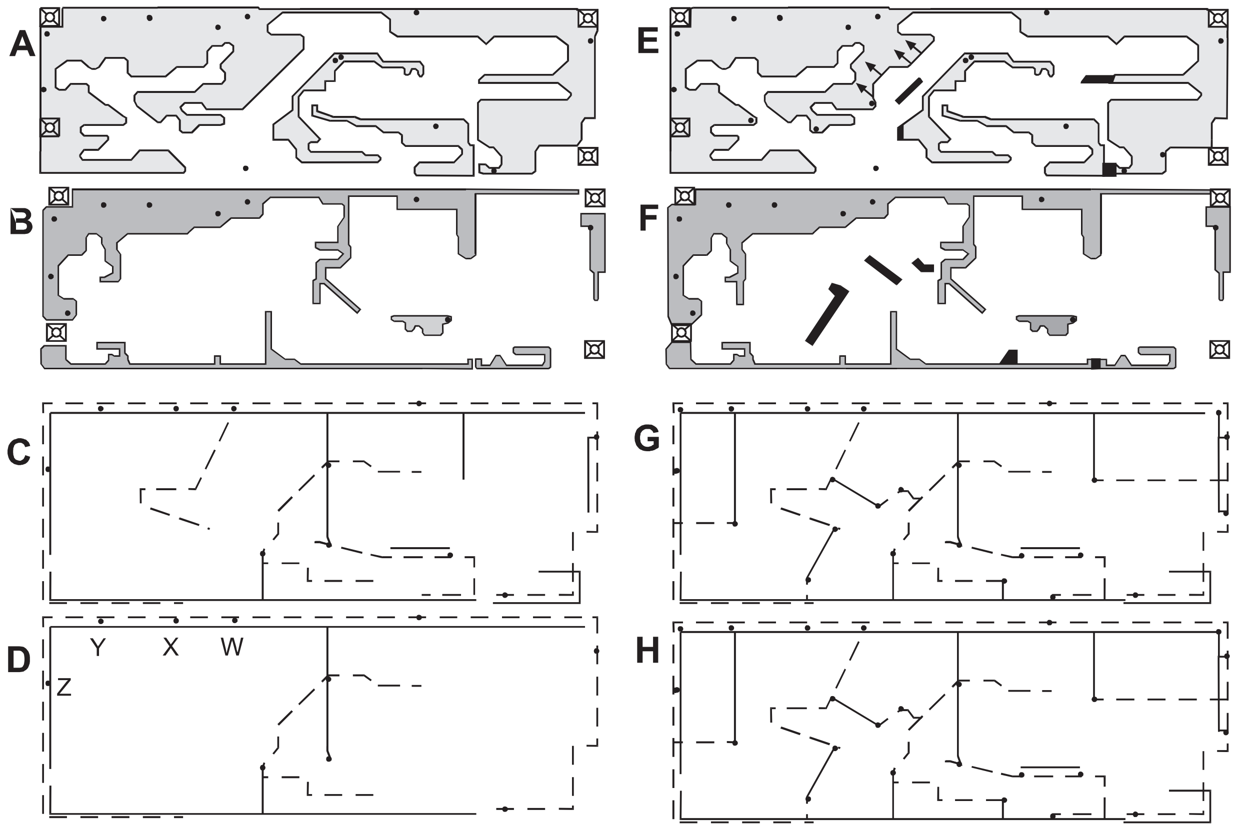

Figure 8. Gridding of Ground Fills and Traces to Form a Ground Plane

In the example in Figure 8, A and B represent the top and bottom sides, respectively, of a simple two-layer board. The +V traces and all interconnects have been deleted, leaving only the ground fill and ground traces, along with the vias between the front and back. Figure 8C is a simple stick diagram of the ground routing for the board. Each stick, or leg, represents the path of the ground conductor, as if the conductor has been shrunk down to a minimum-width trace. The top-side traces are represented by the dashed line, and the bottom-side traces by the solid line. It is easy to see in this diagram that most traces are dead ends. Most traces are connected at only one end. In Figure 8D, most of the single-ended traces have been removed. The result is a sparsely connected pattern that represents how ground is routed over the entire board. Excluding points W, X, Y, and Z in Figure 8D, there is only one path between any two points anywhere on the routing.

In Figures 8E, 8F, 8G, and 8H, the design has been modified very slightly, to achieve a gridded ground. In Figures 8E and 8F, the addition of some traces, shown in solid black, and slightly moving some geometries, as indicated by the arrows, has created an extensive network of interconnections that creates the desired grid. This is shown by the stick diagram of ground in Figure 8G. Closing the gaps around the mounting holes also contributes to the network. No longer are whole traces connected at only one end. Now, they connect at both ends, and form a more complete conductor. Figure 8H shows the density of the grid, which contrasts to the openness of Figure 8D. Also, notice how, in Figure 8H, no traces are dropped because they connect only at one end. Only one trace has this problem, and it is part of a geometry already connected in three other locations. This interconnected network is the goal of gridding ground. The result is nearly as effective as an actual ground plane.

2.2.4 Bypassing and Ferrite Beads

Bypassing between the +V and ground at the microcomputer is critical because the intent is for the capacitance to supply the current used in the device for switching. If the current is not available in the bypassing loop, because of too much inductance, the laws of physics say that the current should come to the lowest impedance, which then is from the leads connecting to the power supply. The distributed capacitance of the power routing becomes the source for the higher frequencies. Thus, the ferrite bead blocks the sourcing of RF current from the power line connection, forcing the microcomputer to live off the current available inside the ferrite bead.

It is of the utmost importance to realize, and always keep in mind, that the power-routing purpose is only to replenish the charge in the bypassing capacitor, and that the bypassing capacitor should supply all currents at or above the oscillator frequency. Keeping RF off the power distribution traces is accomplished using these measures (see Figure 9):

- Use a ferrite bead and a bypassing capacitor (0.1 µF or 0.01 µF), placing the capacitors inside the ferrite bead. Place a 1000-pF capacitor outside the ferrite bead, creating a PI filter. The ground connection for this capacitor should be the microground. However, if there is a lot of noise on this point, the capacitor could couple that noise back onto the +V line.
- The ferrite bead is used only on +V, not on ground. If a through-hole ferrite bead is used, it is mounted with the exposed lead connected to +V.
- Apply the 3:1 length-to-width rule for traces in the bypassing loop, to minimize impedance in this high-frequency path.
- Make the bypassing loops as small as possible in area and length. When tying the bypass capacitors for the oscillator or +V supply, try to extend the microcomputer ground rather than running a trace. Try to run any trace back over (or under) any other segment of the loop to reduce the radiating area when viewed from the top of the board.
- It is acceptable and beneficial to use ferrite beads and the same bypassing values on four-layer boards. The 1000-pF capacitor may not be needed on four-layer boards, but it should be drawn in the initial design, and deleted later if screen-room testing shows that it is not needed.

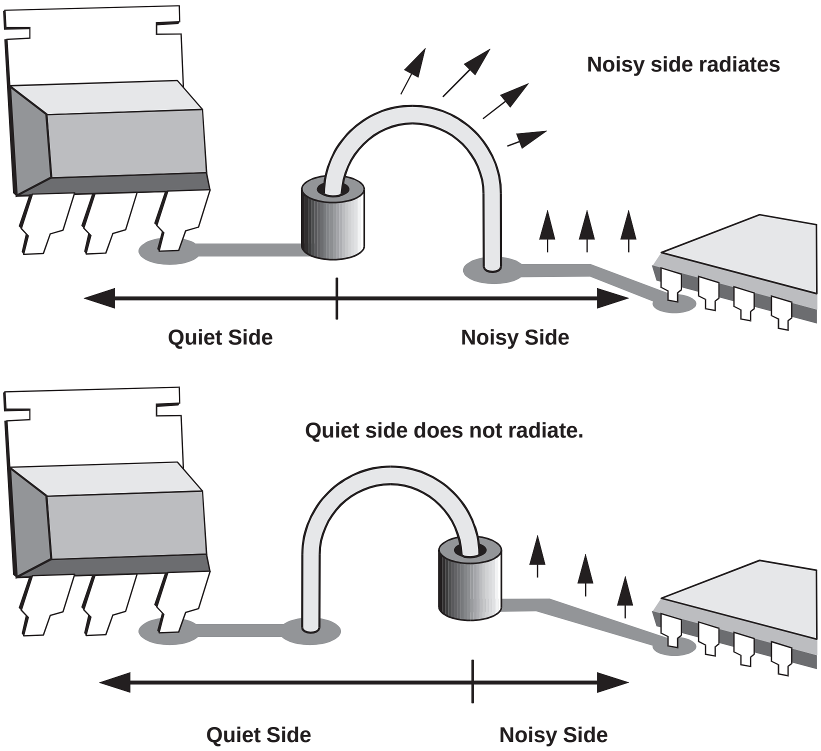

Figure 9. Ferrite-Bead Placement Closest to the Noise Source