Good Reasons to Locate Critical Tracks on Inner Layers of a PCB

Here are several good reasons why you shouldn't put critical tracks on the surface layer of a PCB. Of course, we all have to compromise sometimes, and critical tracks may end up on an outer layer, but do be aware of the compromise that you make when you place them there.

Susceptibility to Electrical Interference from Other External Devices

Radiated signals from other electronic devices, like cell phones, electric motors, computers and their monitors, are all around us. Exposed conductors, such as surface layer tracks on PCBs, receive those signals in just the same way an antenna does. The received signals are added to the real signal as noise, and can prevent the circuit from working properly in its installed location.

Conversely, inner layer conductors are well shielded from the same radiated signals by the solid copper layers on each side of the conductors. Radiation can sneak in at the edge of the board, between the solid layers, but is attenuated by absorption into the copper surfaces before it can penetrate deep into the heart of the board layer. For that reason critical tracks should be routed away from the edge of the board, and a distance of 10x the layer thickness is often recommended as the ideal minimum.

(By the way, tracks also transmit radiation as well as receive it, so the exact opposite can happen. And, as above, the best way to limit the radiation is to position tracks between solid copper planes, which also suppress radiation. As above, radiation can escape through the board edges, and the same 10x layer thickness guideline applies.)

Control of the Factors Which Determine Impedance

The shape and proximity of adjacent conductors are factors that contribute to track impedance value. Conducting surfaces of other boards, cables and chassis parts placed near surface tracks when the PCB is placed in service will change the surface track impedance.

Placing the controlled impedance tracks on inner layers, where the positions of all adjacent conductors is pre-determined, maintains a known environment and preserves the impedance value.

Precise Control of Track Cross-Sectional Shape

The precise cross sectional shape of a track contributes to impedance by determining the charge distribution around the surface of the conductor. Charge tends to concentrate at the sharp corners of a track, so that the associated electric field is strong there and relatively weak at the centers of the flat track faces. Even a slight loss in corner sharpness causes some re-distribution of the electric charge away from the (rounded) corners, hence re-distribution of the electric field occurs, accompanied by a change in impedance.

Field solvers assume either a rectangular or trapezoidal cross section for tracks, as that is the normal result of a high quality etching process. Tracks on inner layers are well represented in this assumption, but plated tracks in outer layers lack the sharp rectangular or trapezoidal profile and, as a result, deviate from the design impedance value.

Precise Control of Track Cross-Sectional Dimensions
The precise cross-sectional dimensions of a track also help determine other dimensions, such as track-to-track spacing, and track-to-plane. All of these dimensions are more accurately and reliably produced for inner (unplated) tracks than for the plated tracks in outer layers. The cumulative effect of the dimensional errors on plated outer tracks can lead to a significant deviation from design value for impedance.

Track Coating Thickness and Profile
Tracks in outer layers are normally coated with a dielectric material, and even the thinnest coating significantly alters the shape of the field and, hence, the impedance of the track.

Design assumptions are made concerning the coating uniformity. For example, that the coating has the same thickness above the track as it does above the dielectric laminate material; the coating has the same thickness above the dielectric laminate between differential tracks as it has above the laminate on the outsides of the track pair; the coating (measured horizontally) on the track sides has the same thickness as the coating on the top surface of the track. In all probability these assumptions are incorrect.

Coating profiles and resulting thickness variations depend on a wide variety of parameters such as coating material (and liquid properties), how it is applied (brush or spray), the direction of application, track and spacing dimensions, and more. Generally speaking, although a coating may be, in some sense, “uniformly” applied, it will redistribute itself before and during the curing process to adopt a “normal” thickness over most of the dielectric laminate surface, an increasing thickness between narrowly spaced tracks (especially between differential pairs, which are normally more closely spaced) and a decreasing thickness on track top surfaces. One can easily imagine liquid coating running off the track tops into the valleys, and capillary action drawing it up the vertical track sides, the more so in narrower valleys. The finished effect has a smooth, rounded surface profile whose thickness varies from design value over the entire board surface, resulting, in turn, in significant impedance deviation from design value.

The profile could, in principle, be taken into account in a field solver if only the designer and builder could predict and define it. That is simply not practical, however, and so critical traces are best confined to inner layers where the profile and thickness of the surrounding dielectric is most accurately characterized.

Polar Instruments Ltd
www.polarinstruments.com
Tel: +44 (0) 1481 253081 Fax: +44 (0) 1481 252476 © 2002 Polar Instruments