Driven by the demand for speed, economy and remote mass production, major changes are taking place in the field of high-speed digital PCB design. We take a look at some common techniques used in the design of multi-gigabit PCBs to achieve right-first-time results with the assistance of modern EDA.

Differential routing

In general, signals are moving from being single-ended to differential, with further structures, such as coplanar waveguides, under consideration. Differential signals, when well implemented, are more robust, as they cancel out any identical noise appearing on both lines. When routing differential pairs, the two travelling signals should be viewed as companions that need to remain in step from beginning to end.

Leading-edge EDA tools treat differential pairs as composite signals rather than two separate nets, automatically considering the requirement to mirror the other signal and adjusting routing accordingly.

Standardised topology

By using standardised topology rather than considering each case separately, common mistakes can be avoided and analysis time reduced. This can be done most effectively by following industry standards, such as the JESD8-15A recommendations for Stub Series-Terminated Logic (SSTL), which allow designers to shortcut the need for detailed analysis of terminations and connection styles. EDA software is needed to constrain the layout to follow these recommendations and to gain competitive advantage, for example by economising on terminators.

Figure 1 (above) - Similarly-coloured sections should have the same trace width and length, and vias should be at identical positions within each section. Bends or lengthening patterns in differentially-routed sections should be balanced to ensure that the signals remain 180 degrees out of phase with each other.

Figure 2 (below) - This defensive series-and-shunt termination style is recommended for some 1.8V double data rate signals. More economical terminations can be used provided designers simulate the proposed topology to ensure signal integrity does not deteriorate too much.

Jitter and eye diagrams

High data rates are forcing the emphasis in signal integrity away from traditional ringing/overshoot analysis to a consideration of the data stream as a whole. Pulses can leave after effects that cause inter-symbol interference in their successors, in addition to problems caused by poor signal integrity or EMC.

This holistic approach is enabled by the use of eye and jitter diagrams in more advanced EDA software. Eye diagrams represent the expected variations in communication channel response due to jitter. A rough definition of jitter is that it is the difference between when an event should ideally occur and when it actually does occur. Distribution of jitter can vary, with common forms being Uniform and Gaussian. Eye diagrams can be generated by both EDA simulation and hardware measurement.

Serialization and its competitors

High-speed bus standards such as PCI Express and HyperTransport can free up significant amounts of board space by reducing internal parallel bus widths. Using these techniques demands careful routing of traces, correct topology and good impedance control.

It advisable to plan the routing style before physical design. Simulation can be used to verify design scenarios and generate constraints that physical design tools can then follow, so the layout is correct by construction. Designers should treat buses, like differential pairs, as composite signals rather than mere groups of nets, and route them as composite items. The latest physical layout tools perform trunk routing, treating buses or sub-buses as composite routing objects. The symmetry that this technique imposes on the bus routing...
means the electrical behaviour can be predicted before physical design begins. Find-and-fix after random auto routing is no longer a realistic option for Gigabit signals.

**Back-drilling**

It is well known that via size has a significant effect on signal integrity. Using vias that penetrate only the required layers and that have pads only on connection layers is one solution, but this is sometimes too costly for mass-production. Back-drilling is an increasing popular technique. Low-cost plated through-hole vias are used initially with the unwanted parts of the vias then drilled out. This means vias have a much smaller parasitic effect on the signal. It is important that EDA solutions support manufacturing processes like back-drilling - electrical models derived from the design data alone cannot necessarily be relied upon to represent the final structure.

**Moving to a higher level**

Good signal integrity can no longer be achieved by post-layout analysis and find-and-fix; a far more integrated, holistic approach is required. With the use of multi-gigabit designs on the steep incline and the demand for bandwidth as great as ever, EDA is moving to a new, higher level.

---

**UV-laser Cutting System For Flex PCBs**

The LPKF MicroLine UV 3000 laser cutting system for prototyping and pre-series production represents the next generation of the MicroLine UV 350D. The system features a high speed linear drive, which delivers greater positioning accuracy (+/- 15micron), higher speed and faster acceleration.

The LPKF MicroLine laser systems are designed for precision cutting, routing, skiving, drilling, cutting pockets, structuring of etch/solder resist, micromachining of ceramic substrates, and more. According to LPKF, the user saves time and money because specific tooling is no longer required.

LPKF Laser & Electronics
Osteriede 7 - 30827 Garbsen
Germany
Tel. +49 5131 7095 324
Fax +49 5131 7095 90
D.Justeschen@lpkf.de
www.lpkf.de

---

**Multilayer Prototype Production For In-House Labs**

LPKF has launched the MultiPress S as part of its family of production tools for complete multilayer prototypes. These allow the production of multilayer prototypes through a three-step process, which includes structuring, pressing and through-hole plating. According to LPKF, the process enables to manufacture and test complete samples in a single day.

An LPKF circuit board plotter is used to structure and drill the different layers of the PCB. The MultiPress S presses and heats the layers (rigid, flexible, HF, etc.), while the company's ProConduct system provides through-hole conductivity without the use of harsh chemicals. The system can be used also for minimal production runs of 5-10 pieces.

LPKF Laser & Electronics
Osteriede 7 - 30827 Garbsen
Germany
Tel. +49 5131 7095 324
Fax +49 5131 7095 90
D.Justeschen@lpkf.de
www.lpkf.de