Designing testability into a PCB

Jason Frels - September 05, 2014

One thing I try to make myself consider when designing a PCB, particularly during layout, is what features I can put in for testing - both for bench testing in R&D as well as during manufacturing test. I try to consider what I could do to speed up validation testing or enable me to better deal with components that were difficult to test in the past. Also, perhaps design in some "undocumented" features to try out some new ideas - board-space and time-permitting, of course.

Test points
If your design gets ICT (in-circuit testing), you will have test-points for the fixture. You will probably try to arrange these to minimize any signal integrity issues. But you can also arrange them to make your design easier to probe. For instance, I have aligned them for a differential probe tip at the interesting end of the signal trace. Or, by adding ground and power test-points in strategic places where I know I'll be probing a lot.

Headers
I put on lots of headers that never make it onto the final bill of materials. I always put in headers for my main voltage rails, just to have an easy place to stick a DMM or connect a cable for chamber
testing, etc. Also, any time I have a JTAG port on programmable logic, I put on a header that will match our programmer. I²C busses can tolerate some SI sloppiness, so I put headers on them. Unused I/O from a CPLD, FPGA, or microcontroller can be routed to a header for debugging or later experimentation.

**LEDs**
I put LEDs wherever I can make them useful. For instance, a DONE signal on a power supply sequencer, to quickly let me know that the thing powered up. Or debug LEDs on FPGAs, microcontrollers, etc., which I can use to debug code issues. They can be stripped off the BOM later, but are really nice when you have a prototype that may not be quite ready for prime-time, so you can quickly see what is happening with it. And, they really look *neato*.

**Test traces**
If there is room, you might just lay out traces that mimic other sensitive traces, so you can experiment with them. Put test-points or connectors on them to connect to scopes, TDRs, signal-generators, etc., so that you can see what your other signals potentially look like. Or, if you have spare clock outputs, run one to a connector; it might be useful for triggering or syncing during testing. I have done this purely for educational reasons when studying crosstalk or other signal integrity issues.

**Undocumented features**
Perhaps, if you have room and time, you might even design and place an entire experimental circuit or extra function that you don’t intend to ship. It might allow you to test out a new idea. I did this once to experiment with some minimum loadings for a power supply. Or it might allow you to have better access for debugging, like a USB port for your embedded device. This does take up board space and design time, though.

I have done lots of other miscellaneous things too, like optional termination schemes, current-sense resistors, trim-pots, terminals for on-board power supplies, etc.

What PCB test tricks and techniques have you employed in *your* designs?

**Also see:**
- [Thermal test chips streamline system development](#)
- [PCB design course & checklist](#)