Application Notes

Application Note 132

Controlled Impedance design for test

Background

TDR (Time Domain Reflectometry) has become the established technique for the measurement of controlled impedance on printed circuit boards. Ideally, trace lengths of approximately 6 inches (as recommended in Standard IPC2141) allow for easy measurement, as there will be a significantly flat region of the trace to allow an accurate average impedance measurement to be made.

Design Considerations

A few simple design considerations will assist fabricators achieve best yields and reduce the cost of your high performance boards

Testing with coupons

The simplest technique for PCB shops is the use of test coupons for verification of controlled impedance PCBs. These are either designed by the customer or added by the fabricator's front end department.

Often a customer will prohibit changes to the coupon. This can require the PCB fabricator to invest in a wide variety of test system probes which must be interchanged frequently, with subsequent reduction in the R&R of the measurement system. It is therefore preferable that the designer and manufacturer collaborate to modify a coupon footprint in order to minimise the number of probe variations required.

Why not use variable pitch probes?

Whilst these probes are readily available, they are expensive and are designed for non continuous operation. Variable pitch probes are less easy for operators to use, and long term high volume use of variable pitch probes can result in poor R&R.

Testing on boards

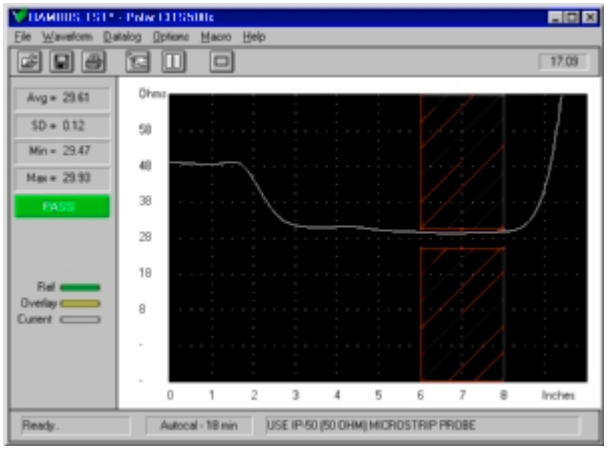
Most testing is performed on coupons. If it is necessary to test traces on boards ideally the in board coupon trace should terminate near the edge and be clearly marked. Also the via holes should be large enough to facilitate manual probing. It is almost impossible to manually TDR test on a microvia! Although the RF contact may be excellent, it is not practical to manually probe such a small contact area, especially if thousands of tests per day are required. If you need to test on actual PCB traces, then remember that the test needs to start at a net end and there must be access to ground next to the signal test point, otherwise it will not be possible to make a good RF connection to the trace under test.

Length of test trace

To obtain repeatable and consistent results in manufacturing, you need to design test traces that are sufficiently long to avoid measurement results being influenced by the effects of probe to trace interconnection aberrations (from the start of the trace) and pulse dispersion (which causes the trace to start to rise before the physical end of the trace.) Although some publications may recommend the specific trace region to be tested (e.g. 50% to 70%) this is not always optimum for short traces.

The optimum trace length on a coupon is around 6 inches (150mm). IPC D317 contains further information on coupon design.

We suggest that you select the test region by choosing the flattest, least disturbed waveform region that falls between any aberrations near the trace start and the climb towards an open circuit at the trace end. Once selected, the same region should be used for all subsequent tests of that type of PCB.

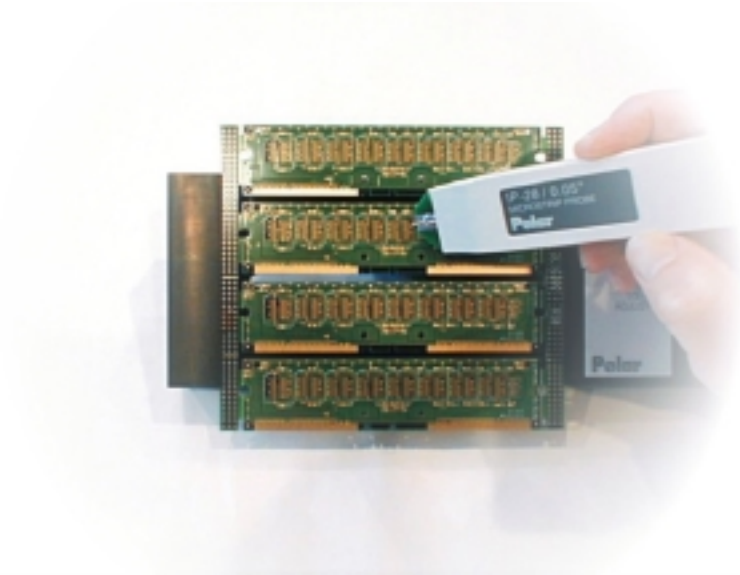


Make it easy for the operator

As you have standardise on a coupon footprint, do ensure that all the contact points on the coupon have the same orientation.

This is beneficial to both manual and automatic testing. With a manual system the operator is not required to repeatedly twist the probe in order to contact the test pads, while in an automatic system, aligning all the points to the same orientation will result in fastest test with best R&R, as the test head needs only X and Y axis positioning.

Where the coupons are built into boards, for example on Rambus® RIMM modules, test point orientation is equally important. Early RIMM designs used test points with ground positions that were 180 degrees rotated at each end of the board. Newer designs have all test points oriented in the same way. If you need to change an early Rambus design please ask us to connect you with the Rambus staff member who can confirm the change authorisation.



Resources

Sample manufacturing data for a coupon containing a wide variety of structures is available for download - (follow the link in the next paragraph). This typical coupon can be modified to suit your process and is designed with test access for both manual and automatic impedance test systems

Download sample test coupon Gerber files: mpcd1345.zip (170kb)

Footprints
A wide variety of probe footprints are available. You can download the latest available footprints for differential and single ended tests by selecting this link: IP Probe Footprints.doc (49kb)

You will see why it is important to standardise on test points so you can avoid holding too many probe variants in stock. If you are just starting to build controlled impedance boards, or if you want to rationalise the types of probes you use, then please email martyn.gaudion@polarinstruments.com who will be delighted to discuss the most popular probe footprints with you.

Modifying designs.
It is important for front end engineers to maintain a good dialog with the original designer of impedance controlled boards. Sometimes small adjustments inline width are necessary to maximise yields, or non availability of particular core or pre preg sizes may require an alteration of the stackup which will, in turn, require re-calculation of the impedance controlled dimensions. To assist in this process the Polar SI6000A field solving impedance design system can goal seek new values and graph sensitivity to changes in build parameters.

More information?
Further information on measuring PCB controlled impedances is available by email from martyn.gaudion@polarinstruments.com

For information on field solving impedance design software please contact: ken.taylor@polarinstruments.com

Polar Instruments Ltd
www.polarinstruments.com
mail@polarinstruments.com

Tel: +44 1481 253 081 Fax +44 1481 252 476