An introduction to the Manufacture and Design of Controlled Impedance PCBs

We produced the first edition of this booklet several years ago in response to many requests for a basic introduction to the manufacture of controlled impedance printed circuit boards (PCBs).
In this second issue, we have added extra material to some sections and introduced a number of new topics to reflect the changes that have occurred in the industry over the last few years.
We have tried to explain the most important concepts and address the most frequently asked questions with the minimum of technical jargon.

Printed versions of this document are available from Polar Instruments or our distributors.

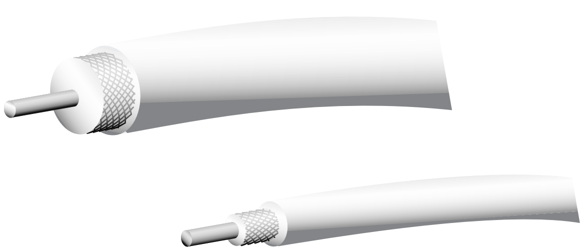
We hope that you find it useful and welcome any comments you may have e.g. areas where you would like more detail, suggestions for new topics, etc. Please email your comments to us at **mail@polarinstruments.com.**

Table of Contents

What is controlled impedance?

The cable that connects the antenna to the television is a common example of a controlled impedance that most of us are familiar with.

This cable may be one of two types:
A coaxial cable consists of a round, inner conductor, separated from the outer cylindrical conductor (the shield) by an insulator. The dimensions of the conductors and insulator, and the electrical characteristics of the insulator are carefully controlled in order to determine the shape, strength and interaction of the electrical fields which, in turn, determine the electrical impedance of the cable.



Instead of a coaxial cable, an antenna may be connected to the set by means of a cable formed of two round wires spaced by a flat strip of plastic. As with the coaxial cable, the dimensions and materials of this wire are carefully controlled so as to give it the correct electrical impedance.

These two cables are examples of different configurations of a controlled impedance but there are many others. In the same way, you will learn that there are many different trace configurations that are used in the PCB industry to achieve a controlled impedance. Controlled impedance PCBs emulate controlled impedance cables, where the coax shield may be represented by a plane, the insulator represented by the laminate, and the conductor is the trace. Just as for the cables, the impedance is determined by the dimensions and materials, and these parameters must be very carefully predicted in design, and then controlled in the manufacturing process to ensure that specifications are met.

Impedance is measured in Ohms (symbol Ω) but is not to be confused with resistance, also measured in Ohms (also with symbol Ω). Resistance is a DC characteristic, whilst impedance is an AC characteristic which becomes important as the signal frequency increases, typically becoming critical for PCB traces at signal components of two or three hundred MHz and above.

Why do we need controlled impedances?

The function of a wire or trace is to transfer signal power from one device to another. Theory shows that maximum signal power is transferred when impedances are matched.

A TV antenna has a "natural" characteristic impedance. At RF frequencies of between hundreds of MHz (megahertz) and GHz (gigahertz), the transfer of maximum signal power from the antenna to the cable requires that the cable impedance match the antenna impedance. Also, the TV impedance must match the cable impedance. Hence, there is a matched system where the antenna to cable to TV impedances all closely match and the maximum amount of signal is transferred from the antenna to the cable and thence to the TV.

Where there is an impedance mismatch, maximum signal power transfer does not occur. Only a portion of the signal power is transferred from the sending device into the receiving device (coax, PCB trace, etc.) The remainder of the signal power is reflected back to the sending device.

Suppose a cable of incorrect impedance links the antenna to the TV. Thus, the TV antenna and the cable impedance do not match, and not all of the TV signal detected by the antenna will pass from the antenna into the cable. Some will be reflected back into the antenna and be re-radiated. Thus the TV will not receive the maximum signal available, and picture quality may deteriorate.

Next, since the cable impedance does not match the TV impedance, not all of the signal conveyed by the cable will pass into the TV. Some will be reflected back into the cable, resulting in a further loss of available signal to the TV.

But this is only the start of the problem. The reflected signal will travel back along the cable to the antenna, where it will encounter the impedance mismatch again, and another partial reflection of the signal will occur. This reflected signal will eventually reach the TV for the second time, somewhat later than the original signal. The TV will simultaneously display multiple copies of the same picture, the original and the delayed reflected signal, like an echo.

Thus the picture (data) has been corrupted by signal reflections caused by impedance mismatches.

Impedances Matched

Impedances Mismatched

The consequences are slight in this example – a degraded TV picture. But suppose that the signal had been carrying bank data, in the form of binary ones and zeros. What could have happened if a zero had bounced back and forth in the cable, and had corrupted your personal account information?

"Sorry, your account is overdrawn"

Controlled impedances on PCB traces

Although we have focused on wire interconnections, exactly the same considerations apply to signal transfer through traces on a PCB. When board traces carry signals containing high frequencies, care must be taken to design traces that match the impedance of the driver and receiver devices. The longer the trace, or the greater the frequencies involved, then the greater the need to control the trace impedance. The PCB manufacturer controls the impedance by varying the dimensions and spacing of the particular trace or laminate.

Any impedance mismatch can be extremely difficult to analyse once a PCB is loaded with components. Components have a range of tolerances, so that one batch of components may tolerate an impedance mismatch, while another batch might not. Moreover, a component's characteristics may change with temperature, so that the problems may come and go. Thus, if changing a component appears to cure a problem, the components may become the suspects instead of the trace. Component selection becomes the solution, and build costs are driven up, while all the time the real fault – trace impedance mismatch – goes undetected!

For these reasons, a PCB designer will specify trace impedance and tolerance, and should work with the PCB manufacturer to ensure that the PCB meets the specifications.

Types of systems that use controlled impedance PCBs.

As recently as 1997, only exotic, high-speed devices of the time required controlled impedance PCBs. These amounted to perhaps 20% of PCBs manufactured. In 2000, around 80% of all multilayer PCBs are manufactured with controlled impedance traces. These would include boards for all types of technologies including

- *telecommunications (analog or digital)*
- *video signal processing*
- *high speed digital processing*
- *real time graphic processing*
- *process control*

Most homes today have numerous low-cost applications of these technologies, for example

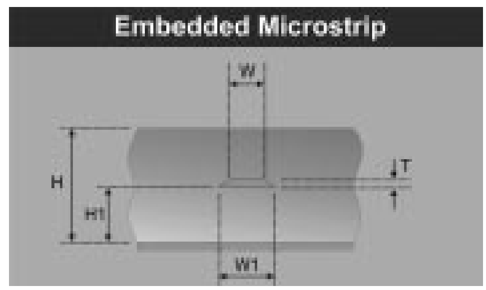
- *Modems, cordless phones, personal radio communicators, analog or digital TV, satellite TV, GPS, radar.*
- *Video games, digital cameras and digital video cameras, DVDs.*
- *Low cost personal computers, CDs, colour printers*
- *Digital TVs, DVDs, Instant Playback TVs*
- *Auto engine control modules.*

Industry and commerce are awash with these same technologies, and the lists are continually expanding. We can therefore expect that in the near future virtually all PCBs will include at least some controlled impedance traces. Controlled impedance has become the norm.

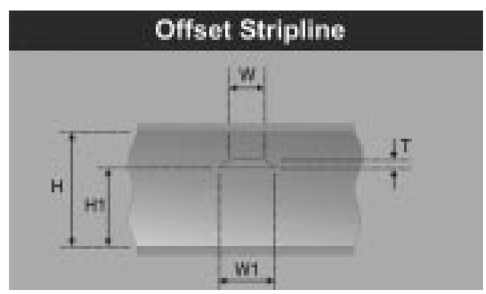
Examples of PCB controlled impedances

These diagrams are examples of some of the many different configurations that PCB designers can use. When you are looking at the stack up of a multiplayer PCB, remember that controlled impedances are shielded by planes and for this reason, you only need consider the laminate thicknesses between the planes on either side of the trace when it is inside the PCB.

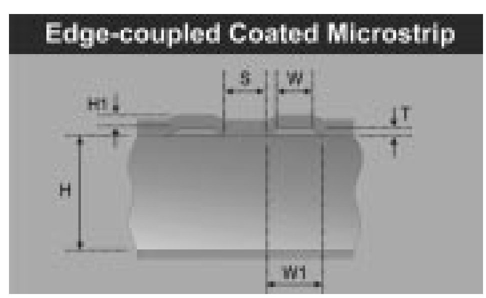
Embedded Microstrip contains a trace sandwiched within the PCB with a plane on one side and laminate and then air on the other.



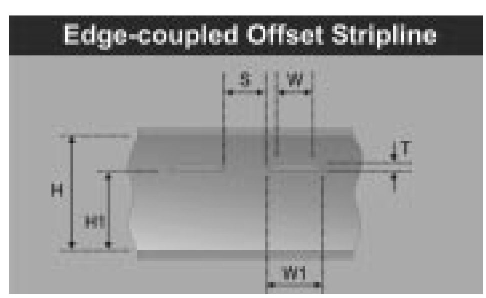
Offset Stripline contains a trace sandwiched within the PCB with a plane on both sides of the laminate.



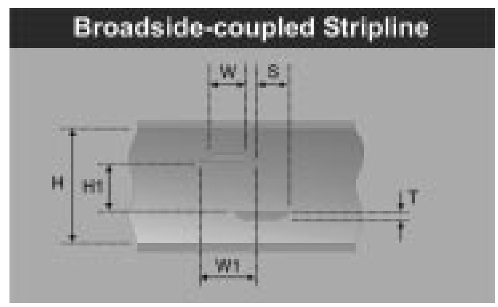
Edge coupled coated microstrip is a differential configuration where there are two controlled impedance traces on the surface, coated by resist and a plane on the other side of the laminate.



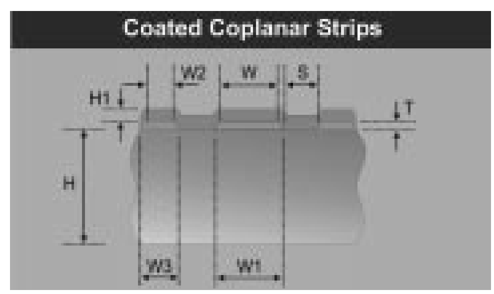
Edge coupled offset Stripline is a differential configuration with two controlled impedance traces sandwiched between two planes. The traces are shown offset, however they could be midway between the planes (2H1+T=H).



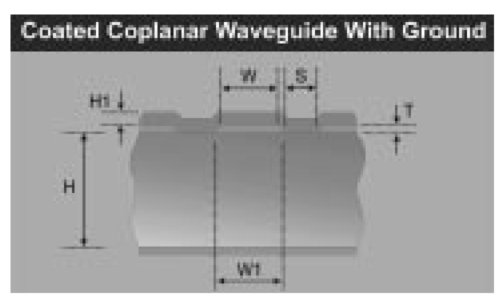
This differential configuration has two traces that are separated by laminate and sandwiched between two planes. Although the diagram shows the traces offset, the manufacturing objective is to have the traces with no offset, i.e. one should be directly above the other. Typically, this configuration is difficult to fabricate.



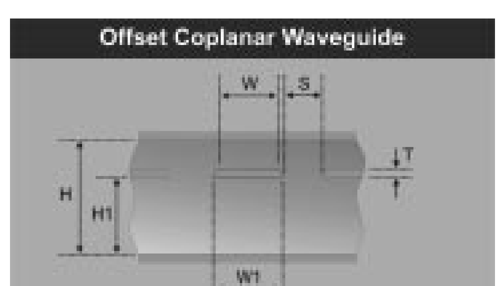
In this Coated Coplanar Strips configuration, there is a single controlled impedance trace with two ground traces of a specified width (W2/W3) either side. All the traces are coated with resist.



The Coplanar Waveguide has a single controlled impedance trace with planes either side (or very wide ground traces), a continuous plane on one side and laminate only on the other side.



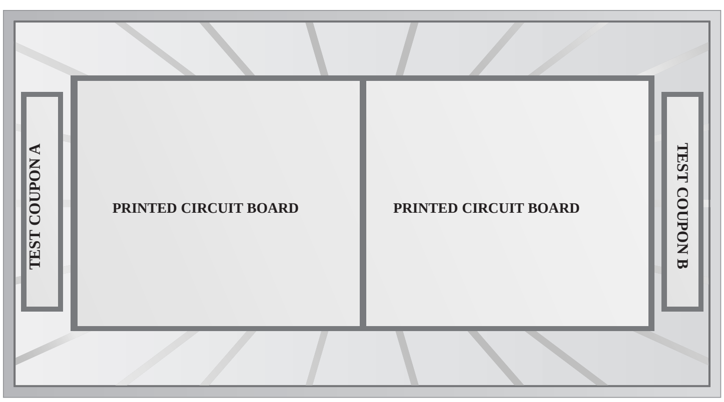
This Coplanar Waveguide is similar to the above configuration except that there are planes on both sides of the laminate as well as a plane on the same layer as the controlled impedance trace.



Manufacturing Controlled Impedance PCBs.

As the operating speed of electronic circuits has increased, so has the need for PCBs to have controlled impedances and the majority of PCB manufacturers are producing them. As described earlier, if the value of controlled impedance is incorrect, it can be very difficult to identify the problem once the PCB is assembled. Since the impedance depends on many parameters (trace width, trace thickness, laminate thickness, etc.) the majority of PCBs are currently 100% tested for controlled impedance. However the testing is not normally performed on the actual PCB but on a test coupon manufactured at the same time and on the same panel as the PCB. Sometimes the test coupon is integrated into the main PCB

Sometimes your PCB customer is not aware that testing is best accomplished using test coupons and you, as the PCB manufacturer, will need to explain the benefits of coupons which include:



Typical Production Panel

All ground and power planes are connected together on test coupon only.

Same aperture codes are used on coupon as on board.

* It is rare for controlled impedance traces to be easily accessible for testing (including a closely situated ground connection).

* Planes are not interconnected on the main PCB and this may lead to inaccurate measurements.

* Accurate and consistent testing results require a straight single trace of 150mm (or longer), often the actual PCB trace is shorter than 150mm.

* The actual PCB trace may have branches or vias which makes accurate measurement very difficult

* Adding extra pads and vias for testing on the PCB will affect the performance of the controlled impedance trace and will occupy space needed for the function of the PCB.

Test Coupons

The typical test coupon is a PCB approximately 200x30mm with exactly the same layer and trace construction as the main PCB. It has traces which are designed to be the same width and on the same layer as the controlled impedance traces on the main PCB.

When the artwork is produced, the same aperture code (D-code) used for the controlled impedance traces is used to produce the test traces on the coupon. Since the coupon is fabricated at the same time as the main PCB the coupon's traces will have the same impedance as those on the main PCB. All planes are included on the coupon and they are interconnected on the coupon only, to ensure that test results are valid. It is necessary to include a void around the coupon on the reference planes so as not to affect the connectivity of the PCB itself if BBT (Bare Board Test) occurs whilst still on the panel.

Usually one coupon is made at the end of each panel to ensure that the coupon is representative of the whole panel i.e. testing the 2 coupons will verify to a high confidence level that there are no differences in trace width, trace thickness, laminate height, etc. over the whole panel.
In fact some customers use the measurement of controlled impedance traces on test coupons on each panel to check the overall quality of PCB manufacture even when the PCB contains no controlled impedances. Since the coupon's controlled impedance depends on all the PCB parameters, it is a very accurate measure of consistency of manufacture without sectioning the PCB.

In addition to the usual PCB specifications, the PCB designer should specify:

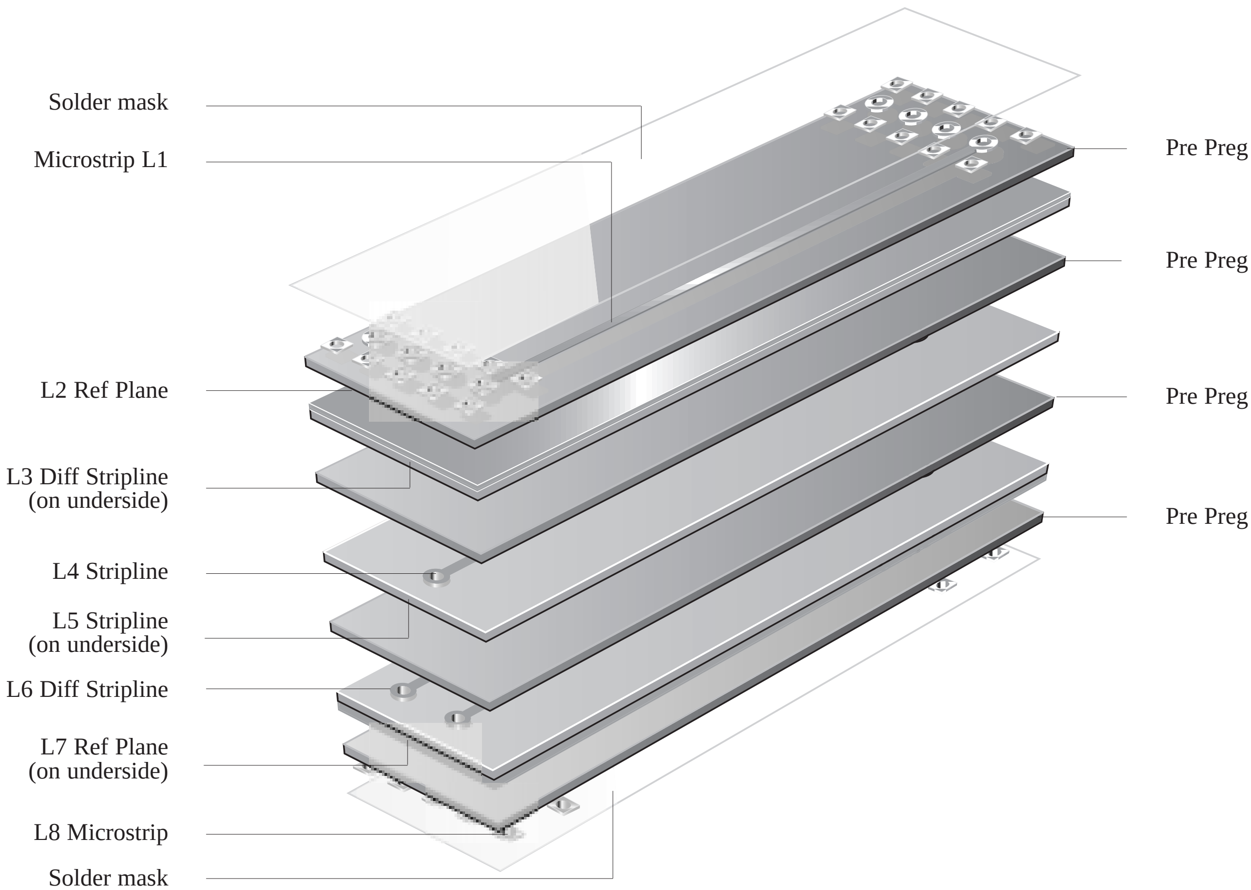
- *Which layers contain controlled impedance traces*
- *The impedance(s) of the trace(s) (there can be more than one value of impedance trace per layer)*
- *Separate aperture codes for controlled impedance traces e.g. 4 mil non controlled impedance trace and 4 mil controlled impedance trace.*
- And either:
  1. the width (w) of the controlled impedance trace
  or
  2. the laminate thickness (h) adjacent to the controlled impedance trace

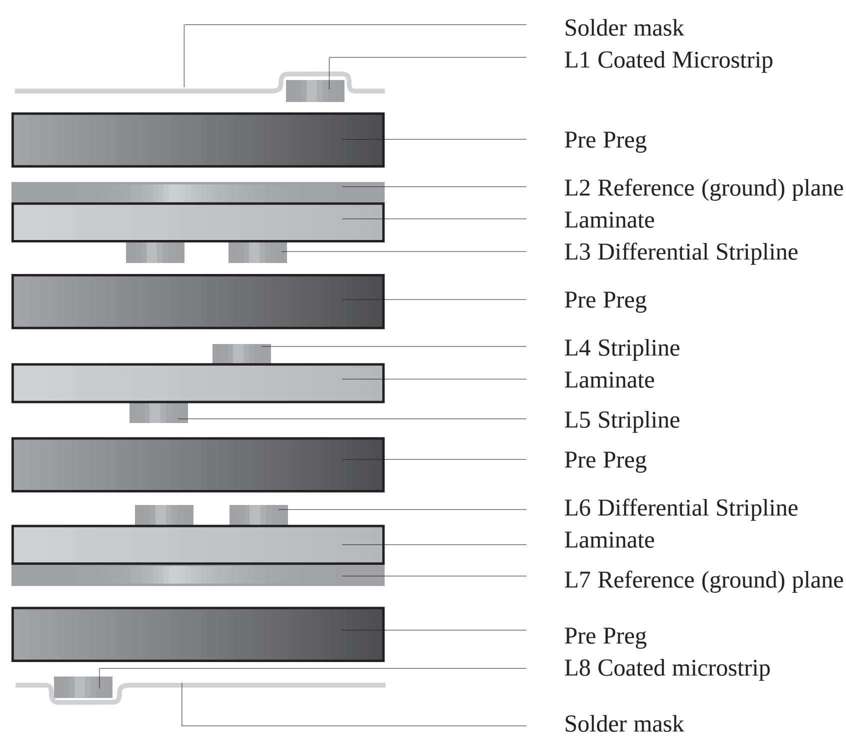
In case 1, where trace width (w) is specified, the manufacturer will adjust the laminate thickness (h) to give the correct value of impedance.

In case 2 where the laminate thickness (h) is specified, the manufacturer will adjust the trace width (w) to achieve the value of impedance.

Some configurations (differential, coplanar) may have more than one parameter that can be varied to obtain the specified impedance.

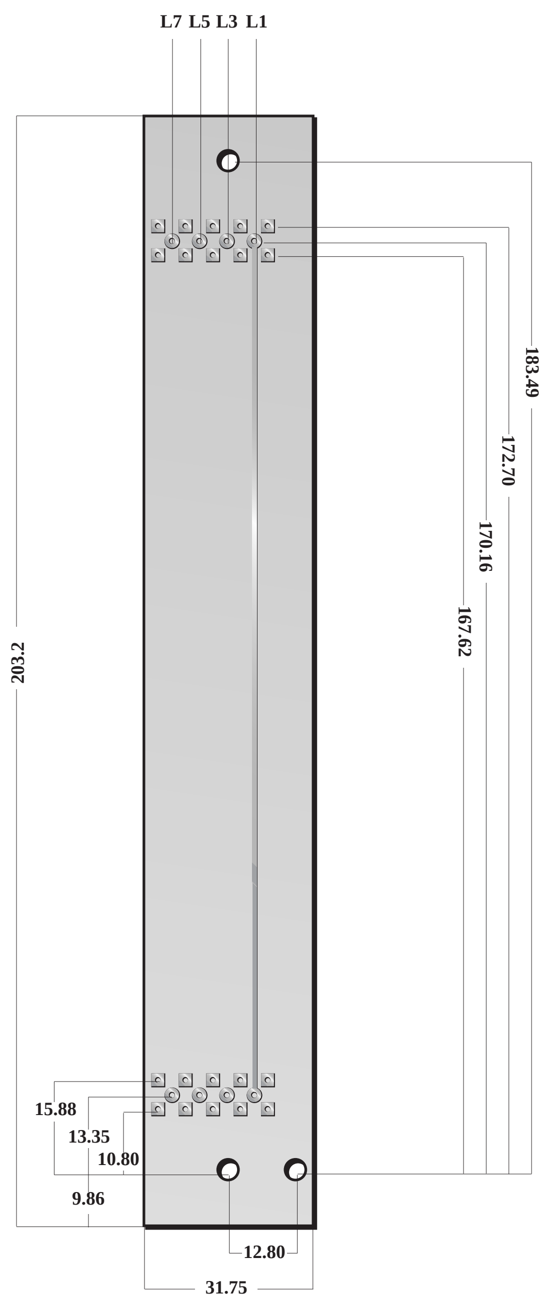
Exploded view of a test coupon

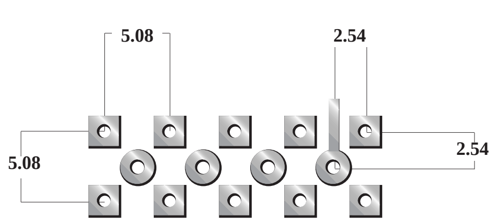




Exploded view of a test coupon

Typical Test Coupon





not to scale

Note that you can download Gerber files for a typical impedance test coupon from www.polarinstruments.com.

Typical Test Coupon
(Reference IEC draft, addition to IEC326-3)

1 Dielectric separation will replicate impedance structure on printed boards.

2. Test connection holes shall be plated through to access all inner layer test conductors.

3. Square pads identify plated through hole connections to access all ground/power reference planes.

4. Conductor widths will replicate critical conductors on each impedance layer.

5. Via holes to be added as required.

6. Cross hatching to be added to outer layer as required.

7. Two coupons per panel. These to be individually identified with letter A & B respectively.

8. Job No. + Datecode to be added as per customer requirements.

9. All planes to be interconnected (on test coupon only).

Capacitive loading

To minimise capacitive loading during test, you should minimise the size of pads and vias on coupons, especially for high impedance traces. Although this standard coupon design shows pads at both ends, you are likely to obtain better test results on high impedance traces if you only place a pad at one end.

Calculating the value of impedances using Field Solvers

A few years ago, you could use simple published equations to obtain the nominal values of trace dimensions for a particular impedance and they were reasonably effective for line widths and spacing above 15 mil (thousandths of an inch). However these simple equations are only approximations and do not give accurate results for line widths used on current technology PCBs.

You will need to use Field Solver software for calculation of controlled impedances. Their effectiveness is enhanced if they offer "goal seeking" that lets you enter the desired impedance and the field solver calculates the trace dimensions.

Polar Instruments produces Field Solver software specifically for the prediction of PCB trace impedances based on trace configuration, trace geometry and material characteristics. You can download free, functioning demonstration versions of the software from our website at
www.polarinstruments.com

Characterising your manufacturing process

Using Field Solver software is a good starting point for obtaining nominal values of trace width (w) and laminate thickness (h) to obtain specific values of impedance. However you will need to produce test panels containing many coupon designs, with different trace widths, different configurations (Stripline, Microstrip, Embedded Microstrip) and different layer structures with different laminate / pre-preg thicknesses.

Ideally you will produce standard coupons (see suggested design) and each coupon can contain a variety of different impedances. After manufacture of the test panels, you will then need to measure the actual values of impedance and see how they correlate against theoretical values.

Laminate suppliers will provide you with lists of Er (dielectric constant) for different core constructions, typically FR-4 has Er=4.2. If you use preferred core material, you will be assured of consistent performance characteristics. By constructing a table of results comparing the measured values with the calculated values, you will see the variance between your process and the theoretical calculations. You can then remake the test panels, altering (w) and / or (h) to obtain the "exact" design values of impedance. After several iterations, you will have an understanding of your process that allows you in most cases to take the designers' requirements and convert them into values that suit your process and produce boards whose impedances are centred around the specified nominal impedance to maximise yield.

It is also useful to microsection some of the coupons to verify the actual dimensions of the traces compared to the nominal values. These measured dimensions can be used in the equations to calculate an impedance value from the actual dimensions, adding a third column to your table of results.

We should mention that the presence of solder resist affects the impedance of surface microstrip and you should include this in your characterisation process.

Impedances typically range between 40 ohms to 120 ohms. The higher impedances are more difficult to control since they typically have narrower traces and will be relatively more affected by the exact etch process (i.e. since the impedance is inversely proportional to the trace width and thickness, as traces become very thin, the relative effect of the etching process will have a greater effect on their width and profile and hence, impedance).

The following relationships will give you an idea of how impedance depends on dimensions, however please remember they are only approximations for fine line widths:

- Impedance is inversely proportional to trace width
- Impedance is inversely proportional to trace thickness
- Impedance is proportional to laminate height
- Impedance is inversely proportional to the square root of laminate Er

Measuring controlled impedances

Polar Instruments designs and manufactures rugged, manual and robotic Controlled Impedance Test Systems specifically for PCB measurements in the manufacturing environment. Polar is a world leader supplying instruments to premier PCB manufacturers around the globe.

Impedances can be measured using:

- **A Network Analyser**
- **A laboratory Time Domain Reflectometer (TDR)**
- **A Controlled Impedance Test System (employing TDR techniques)**

Both Network Analysers and laboratory TDRs are highly complex and sophisticated laboratory instruments that typically need to be operated with great care even by a skilled engineer. A Controlled Impedance Test System (employing TDR measurement techniques) that is specifically designed for measuring controlled impedances on PCBs offers the optimum solution.

A TDR applies a very fast electrical step signal to the coupon via a controlled impedance cable (and preferably a matching impedance probe). Whenever there is a change in impedance value (discontinuity), part of the signal power is reflected back (as discussed earlier) to the TDR instrument which is capable of measuring this reflected signal.

The time delay between the transmitted pulse and the receipt of the reflected signal is proportional to the distance of the discontinuity. The magnitude of the reflected signal is related to the value of the discontinuity.

From this data it is possible to graph impedance versus its position on the test coupon. This is typically achieved using software to control the TDR and to acquire and process the data which it produces.

A TDR specifically appropriate for the measurement of PCB controlled impedance in a manufacturing plant should be able to:

- *Be operated reliably and conveniently in the normal plant environment by a non-technical person with minimum training requirement.*
- *Offer a degree of test automation for high throughput lines.*
- *Produce easy-to-understand results in the form of graphs of impedance versus distance over the length of the test coupon.*
- *Indicate and log unequivocal Pass/Fail results for each device tested.*
- *Datalog results and produce reports suitable for presentation to the customer.*
- *Store test files which contain the specifications for each type of coupon produced, and automatically set up the TDR.*

Using airline standards

Precision airlines are accepted as the "standard" for controlled impedances. They consist of two accurately machined concentric tubes where all of the dimensions are very tightly controlled, terminated with a suitable connector. TDRs used for impedance measurement are precision measurement instruments and need to be regularly calibrated. Reference air line standards are used for TDR calibration and are available in a variety of standard impedances (typically 28Ω, 50Ω, 75Ω and 100Ω).

Precision air lines are a high cost item and an acceptable alternative for less critical applications is a set of precision semi rigid cables calibrated against air lines.

Air lines are made traceable to National Standards (NIST, NPL) by a precision metrology technique called air gauging which allows the bores of the air line to be measured and the impedance calculated using a standard formula.

Differential configurations

Many modern designs use a differential pair of traces between components. When compared to a single trace, differential configurations are less susceptible to interference from adjacent traces and generate less interference.

To be effective, they need to be matched i.e.:

- *Both traces should have the same dimensions and spacing to adjacent traces and planes*
- *The traces should be as close as possible to each other as the manufacturing process allows*
- *The spacing between the two traces should be constant*

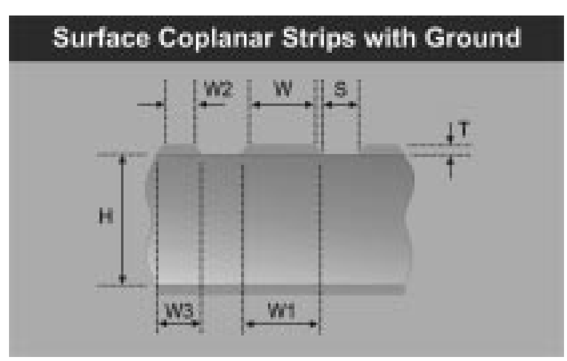
The value of the controlled impedance depends on the trace separation as well as the dimensions of the individual traces and when you are measuring them, you will need to make a differential measurement.

Typically the differential measurement will be slightly below twice the value of the impedance of each trace e.g. if you measure each individual trace of a 100Ω differential pair, they are both likely to read around 53Ω or 55Ω.

Coplanar configurations

Coplanar configurations have become increasingly popular in the past few years and are widely used for Rambus™. One of the benefits of surface coplanar configurations is that they extend the operating frequency of FR4 laminate which loses performance above 2Ghz. In surface coplanar, most of the field between the trace and plane is through air rather than the laminate and so the loss caused by the laminate has less effect at high frequencies.

There are many variants of coplanar configurations and you should also be aware that coplanar differential traces can be fabricated.



This diagram shows just one of the many coplanar configurations .

Typical Questions and Answers

Q. My customer says I need to test their PCBs at 900MHz. Can I do this with a TDR based test system?

A. Yes, a TDR based impedance test system is suitable for testing over a wide range of frequencies. The parameters that determine impedance (laminate Er) do not vary significantly below 3 to 4GHz. So it is unnecessarily expensive and time consuming to do a single frequency test using a network analyser.

Q. My customer does not specify that I measure controlled impedances, what should I do?

A. Your customer may think that by specifying the dimensions, the traces will automatically have the correct value of impedance. As explained earlier, each manufacturing process requires characterisation to ensure that the process is matched to nominal values produced by Field Solver calculators. Your customer will not be satisfied if their PCBs fail because the final value of controlled impedances are incorrect. Work in partnership with your customer and help them understand the need for test.

Q. How do I calculate the dimensions for controlled impedances on inner layers on my stack up?

A. You can ignore any layers that are beyond the planes either side of the trace being calculated. You only need to consider the laminate thicknesses either side of the trace to the nearest planes on both sides. You can think of this as the plane forming a shield either side of the trace.

Q. Why are all of the impedances measurements on my coupon are wrong but the dimensions agree with the Field Solver?

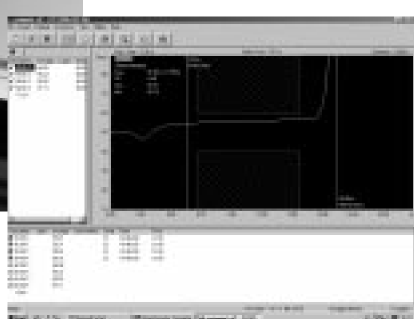
A. You have probably forgotten to connect all of the planes to each other. This is necessary to obtain the correct values. Remember that this should be done on the coupon only and you should leave a void around the coupon to avoid these interconnections affecting bare board test if the coupon is attached.

You can read and download a wide range of Application Notes from our website at **www.polarinstruments.com.**

Controlled Impedance Test Systems (CITS)

The Polar Instruments CITS controlled impedance test system has become the industry standard for use by non technical operators in manufacturing environments.
It is also widely used by CEMs to verify conformance of incoming PCBs.

CITS uses TDR techniques to measure controlled impedances and it automatically reports when a measurement is outside the specified tolerance. It automatically processes the data to produce a simple display of impedance versus distance.



All results and system set up data are automatically datalogged. Data can be easily exported to a third party database (eg for statistical process control). Test Reports can be produced suitable for supply to customers.

High accuracy is assured with instruments that run 32 bit software as they are factory calibrated against precision reference airlines at 28, 50, 75 and 100 ohms, all traceable to National Standards. Users report excellent gage R&R results.

Robotic Impedance Test System (RITS)

The Polar Instruments RITS robotic impedance test system is used for medium to high volume testing of controlled impedance PCBs. The measurement system is based on the industry standard CITS with robotic automation speeding the throughput and offering unparalleled accuracy, reliability and consistency of test.

Impedance calculation using SI6000 Field Solvers

SI6000 is the result of feedback from the hundreds of customers who use Polar CITS25 for controlled impedance design. New Field Solvers in the SI6000A allow you to graph impedance against various PCB parameters.

Using Excel 97 or 2000 as a powerful and convenient user interface, the SI6000A can be used as supplied to goal seek for various designs or graph Zo against a range of parameters.

The SI6000A supports a huge number of popular impedance controlled structures and allows you to fully evaluate their behaviour.

You benefit by producing impedance controlled boards with better yields and as a result have fewer board turns before ramping up production.

You can download an evaluation copy from our website

www.polarinstruments.com