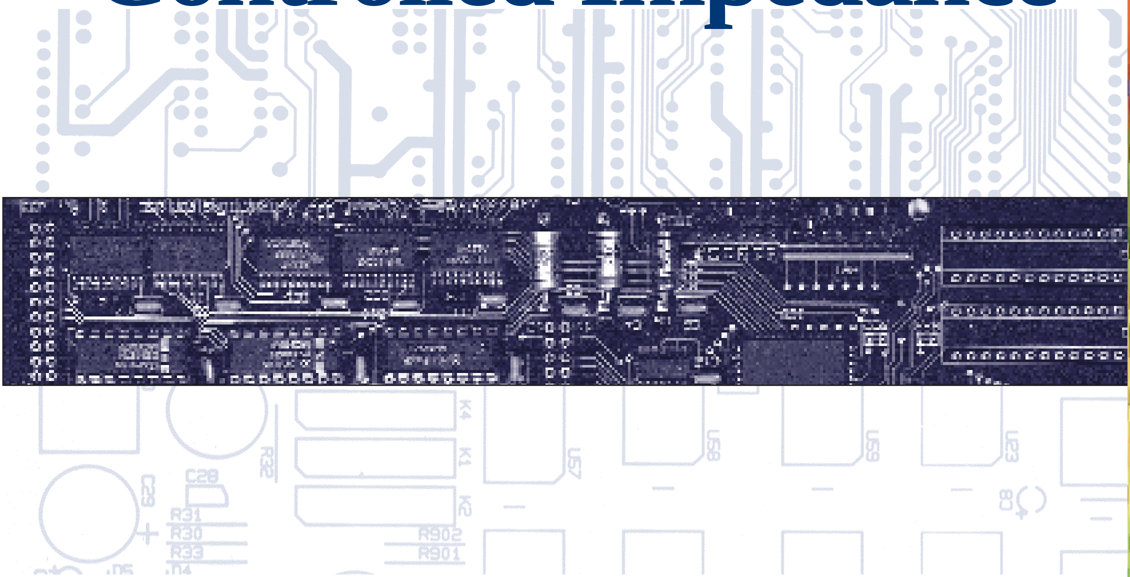
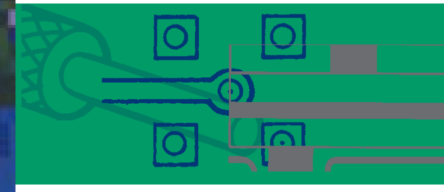
Controlled Impedance



An introduction to the Manufacture of Controlled Impedance P.C.B.'s



Polar

Introduction

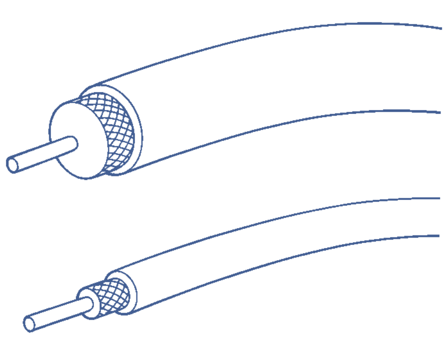
Over the past few years, we have received many requests for a basic introduction to the manufacture of controlled impedance circuit PCB's from companies who wish to start making them and this booklet is the result. What we have tried to do is to explain the important concepts and use the text to answer the most frequently asked questions.

We hope that you find it useful.

We would welcome any comments you may have on this booklet e.g. areas where you would have liked more detail, suggestions for new topics, etc. any feedback would be appreciated. Please fax or e-mail your comments to the address on the back page.

What is a controlled impedance?

The coaxial cable that connects the aerial to the television is an example of a controlled impedance that most of us are familiar with. This special cable consists of an inner conductor, separated by an insulator from the outer conductor (the shield) where *the dimensions of the cable and insulator are carefully controlled* to ensure the cable has the desired impedance.

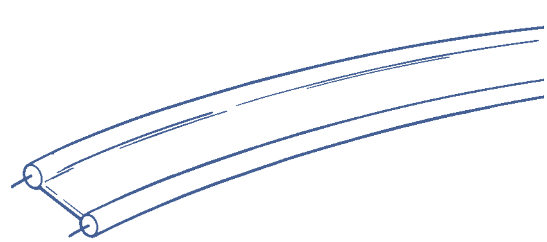


The spacing between the inner conductor and the shield and the material used as the insulator between the two conductors determines the high frequency impedance (measured in ohms) of the cable. In the diagrams above, where the inner conductor is the same, the cable with the larger outer diameter has a higher impedance.

Don't confuse the term impedance with resistance. Resistance is valid for low frequencies and DC whereas impedance in this context is used for high frequencies (e.g. hundreds of MHz).

You will have seen that indoor aerials are sometimes connected to a TV with two wires separated by a flat strip of plastic. This is another type of construction resulting in a controlled impedance i.e. again as with the coaxial cable, the dimensions are carefully controlled to give the correct impedance.

Controlled impedance PCB's are effectively a simulated coaxial cable where the coax shield is represented by a plane, the insulator is the laminate and the coax inner conductor is represented by the trace.



Why do we need controlled impedances?

The function of a cable (or a trace on a PCB) is to transfer power from one device to another. Theory shows that maximum power is transferred when the impedances are matched. Your TV aerial has a “natural”, characteristic impedance and the coaxial cable impedance is designed to match the aerial’s impedance. The input impedance of the TV is also designed to match the coaxial cable’s and aerial’s impedance. Hence there is a matched impedance system (aerial to coaxial cable to TV).

The situation is exactly the same on PCB’s which are operating at high frequencies (these can be analog or digital systems). Designers need to control the impedances of some critical high frequency traces to ensure that one circuit couples the maximum energy into the other circuit. The longer the trace and the higher the operating frequency, the greater the need to control the impedance. The manufacturer of the PCB controls the impedance by varying the dimensions of the particular trace.

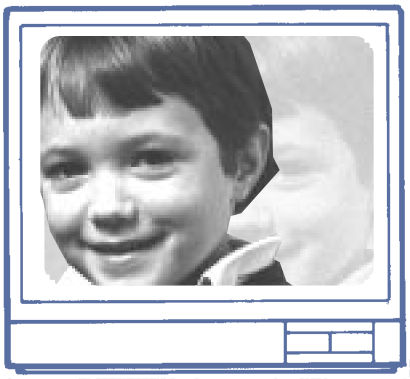
The specific effects of having the wrong impedance depend on the particular circuit but can include:

- ❑ **Low gain in an amplifier**
- ❑ **Excessive electrical noise**
- ❑ **Random errors in digital systems (due to reflections from an impedance mismatch).**

The above faults can be very difficult to identify once a PCB is loaded with components.

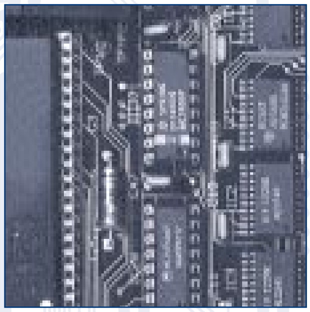
Components on PCB’s have a range of tolerances so that one batch of components may tolerate an impedance mismatch more effectively than a different batch and effectively mask an incorrect impedance trace. In addition, a component’s parameter may alter with temperature so that it can absorb the mismatch at room temperature but random faults occur as it becomes warm. For this reason the PCB designer will specify a value and tolerance of impedance for specific traces and will **rely on the PCB manufacturer to conform to the specification.**

Impedances Matched



Impedances Mismatched

Types of Controlled Impedances on P.C.B.'s



Typical PCB impedances are specified between 40 ohms to 120 ohms and either consist of a trace sandwiched between 2 planes or a trace above 1 plane. The trace and the plane(s) form the controlled impedance.

P.C.B.'s that contain controlled impedances are typically used in:

- **Telecomms**
- **High speed backplanes**
- **High speed computers including PC's, PCI bus**
- **Radar**
- **Military applications**
- **Video signal processing**
- **Real time graphics**

This list is continually expanding as the general operating speed of electronics increases. It is likely that in a few years, controlled impedances will be a frequent requirement rather than a specialist need as at the present.

In the majority of cases the PCB will be multilayer and the controlled impedance can be constructed in several ways.

The value of the impedance is determined by:

- **The width and thickness of the trace**
- **The height of the core or pre-preg either side of the trace**
- **The "dielectric constant" of the core or pre-preg material**
- **The particular configuration**

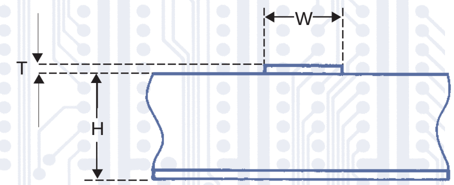
Surface Microstrip

(Configuration 1)

The trace is on the outer layer of the PCB above a plane. The table shows some typical values of impedance and dimensions. The presence of solder resist on the surface affects the impedance (typically reducing it by 1 or 2 ohms).

W	T	H	Impedance
110	1.4	62	50
53	1.4	62	75
26	0.7	62	100

All dimensions in mils (0.001").



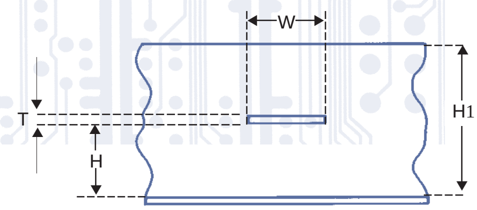
Embedded Microstrip

(Configuration 2)

The trace is embedded within the PCB, with a plane on one side only.

W	T	H	H1	Impedance
53	1.4	31	62	50
12	0.7	15	31	75
5.4	0.7	15	31	100

All dimensions in mils (0.001").



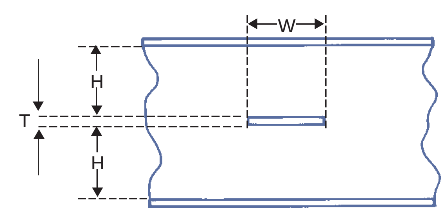
Stripline

(Configuration 3)

The trace is symmetrically sandwiched between two planes.

W	T	H	Impedance
48	0.7	62	50
19	0.7	62	75
7	0.7	62	100

All dimensions in mils (0.001").



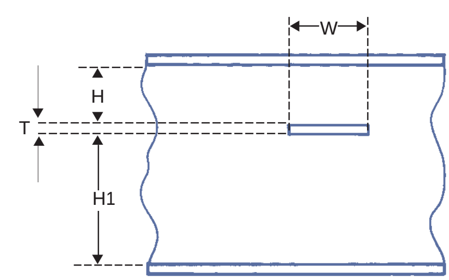
Offset Stripline

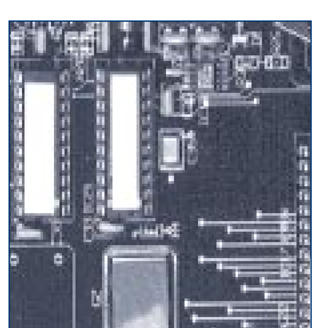
(Configuration 4)

The trace is sandwiched between two planes but is not central. This is also sometimes called "Dual Stripline" because there may also be other controlled impedances in another layer sandwiched between the same two planes. However the controlled impedance is between one trace and the two planes. Any trace crossing between the controlled impedance trace and the plane should do so at right angles to minimise its effect on the controlled impedance (i.e. to minimise the area of overlap).

W	T	H	H1	Impedance
31	0.7	31	62	50
7.5	0.7	15	62	75
3.2	0.7	15	62	100

All dimensions in mils (0.001").



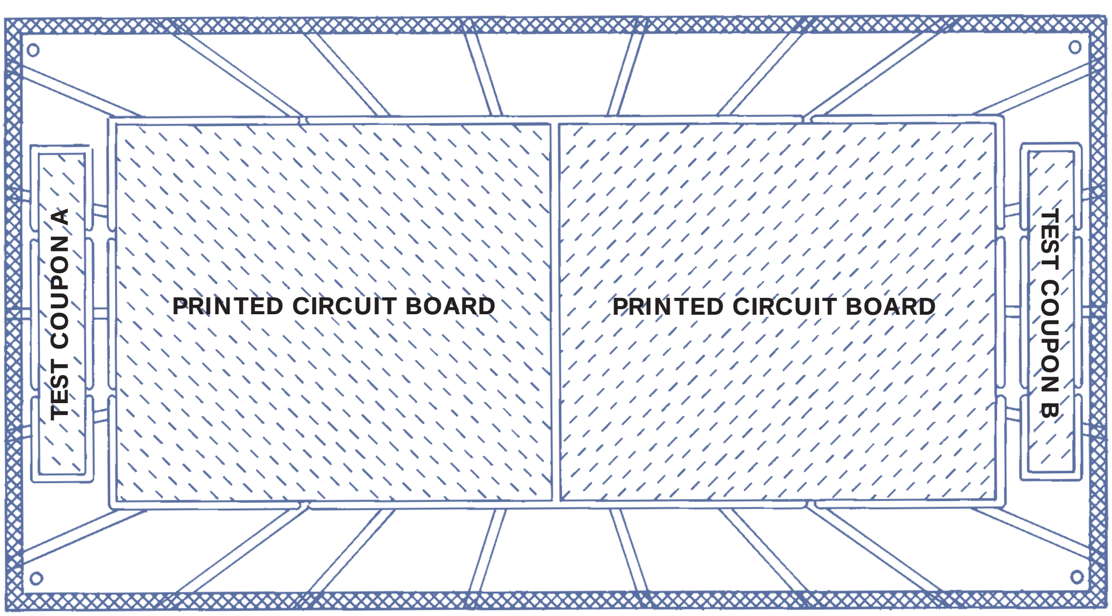


Manufacture of Controlled Impedance PCB's

As the operating speed of electronic circuits increases, so will the requirements for PCB's with controlled impedances and it is likely that in a few years, the majority of PCB manufacturers will produce them. As described earlier, if the controlled impedance is incorrect, it can be very difficult to identify the problem. Since the impedance depends on many parameters (trace width, trace thickness, laminate thickness, etc.) the majority of PCB's 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:

- ❑ **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.**



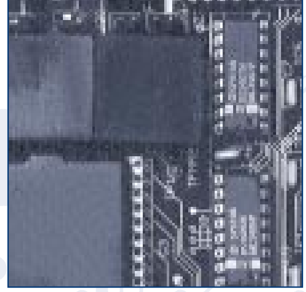
Typical Production Panel

NOTES

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

Same aperture codes are used on test coupon as on board.

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 will be 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 check that there is no change in performance over the 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 although the PCB contains no controlled impedances. Since the controlled impedance depends on all the PCB parameters, it is a very accurate measure of consistency of manufacture without sectioning the PCB.

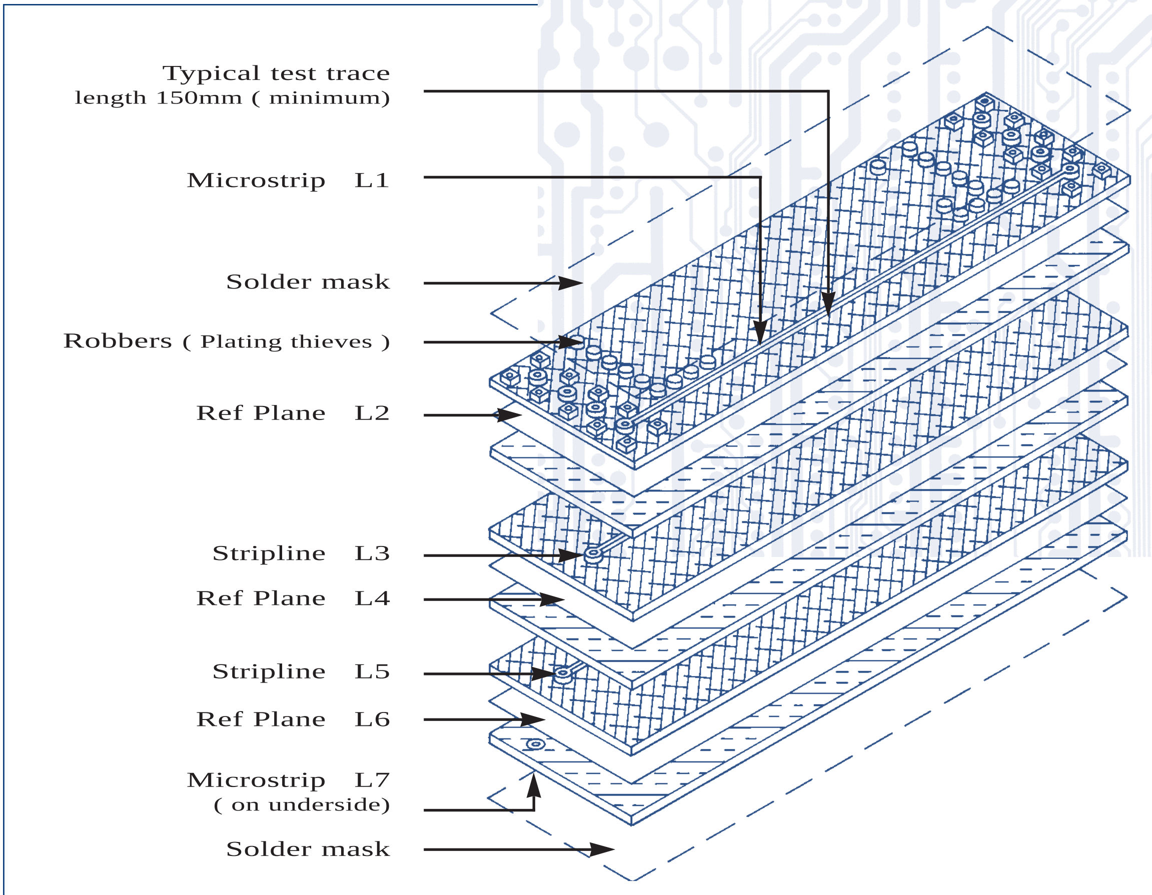
In addition to the usual PCB specifications, the PCB designer will typically specify:

- **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)**
- **Either:**
  1. the width (w) of the controlled impedance trace
  or
  2. the laminate thickness (h) adjacent to the controlled impedance trace

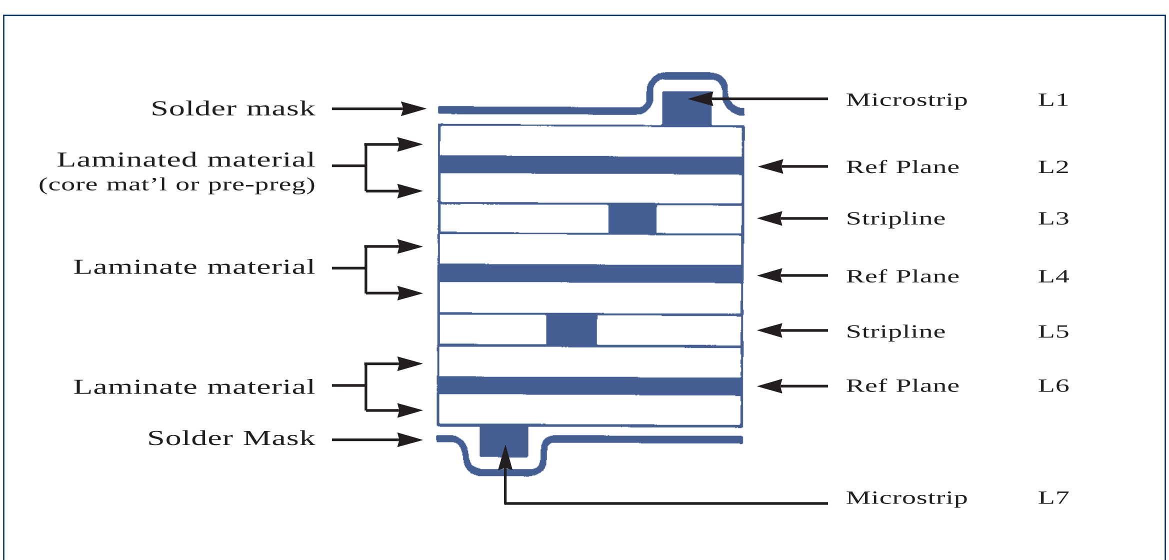
In case 1, where trace width (w) is specified, the manufacturer will adjust the laminate thickness (h) to give the correct value of impedance.
In case 2 where the laminate thickness (h) is specified, the manufacturer will adjust the trace width (w) to achieve the value of impedance.

You may be wondering why the PCB designer cannot specify the trace width and the laminate thickness so that the manufacturer will automatically produce the correct value of impedance. Unfortunately all PCB manufacturing processes are unique and even if two manufacturers use exactly the same dimensions, the value of impedances could differ by 5% or even 10%. It is critical for a PCB manufacturer to characterise his manufacturing process for controlled impedance, i.e. to know the results from your particular process in relation to achieving specified values of controlled impedance from different build configurations.

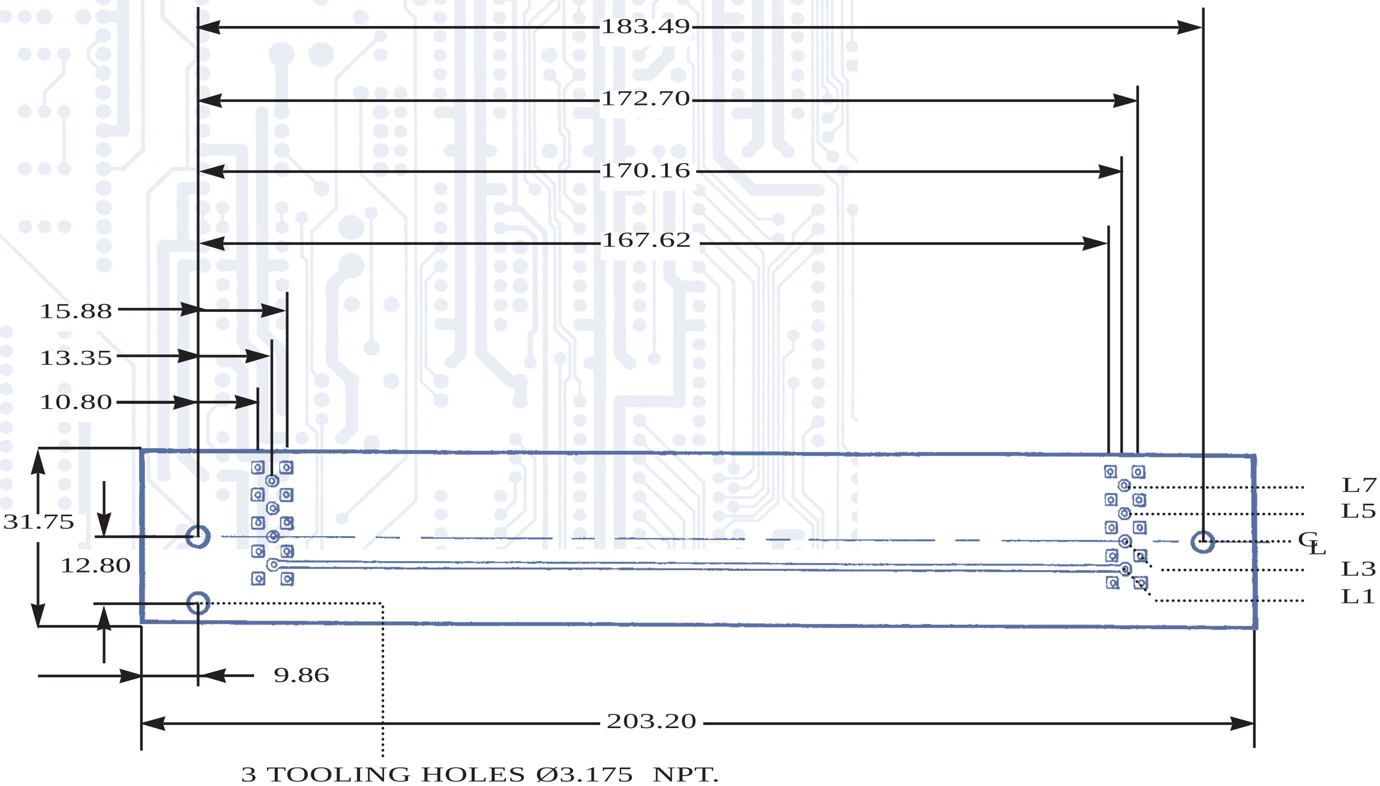
Exploded view of Typical Test Coupon

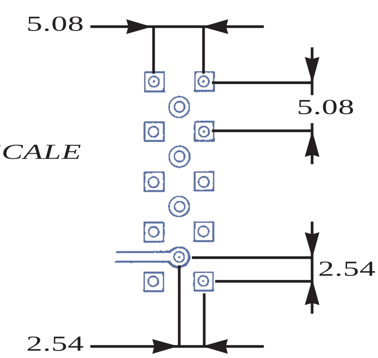


Section Through Test Coupon



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





NOT TO SCALE

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. + Datacode to be added as per customer requirements.

Characterising your process

Equations (e.g. using an Impedance Calculator disk described later) are a good starting point for 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=4.6. If you use preferred core material, you will be assured of consistent performance characteristics. Currently, laminates of 5, 8, 10, 12, 14, 20 and 28 mils are readily available and frequently specified. Sizes of 3, 4 and 6 mils are beginning to be specified in newer PCB's.

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).

Impedance
is inversely proportional to trace width $Z \propto \frac{1}{w}$

Impedance
is inversely proportional to trace thickness $Z \propto \frac{1}{t}$

Impedance
is proportional to laminate height $Z \propto h$

Impedance
is inversely proportional to the square root of laminate Er $Z \propto \frac{1}{\sqrt{Er}}$

Measurement of Controlled Impedance

Impedances can be measured using a Network Analyser or TDR (Time Domain Reflectometer). Both of these are highly complex and sophisticated laboratory test instruments and traditionally need to be operated by an experienced engineer. Various PCB Standards (e.g. IPC-D-317A) recommend the use of TDR as the preferred method of test.

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

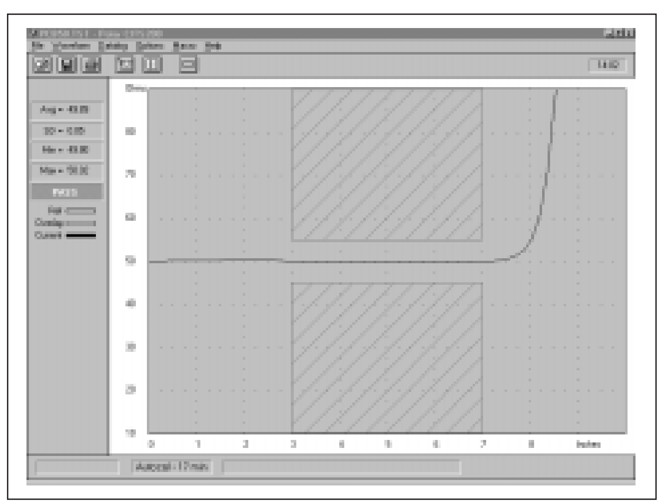
The time delay between the transmitted pulse and receipt of the reflected pulse is determined by the distance of the discontinuity from the instrument. The amplitude of the reflection is related to the value of the discontinuity.

From the above, you can see that the TDR instrument is able to plot a graph of impedance (amplitude of the reflection) versus distance (time between transmission and receipt of the reflection) over the length of the test coupon. This is typically achieved by software running on a PC (Personal Computer).

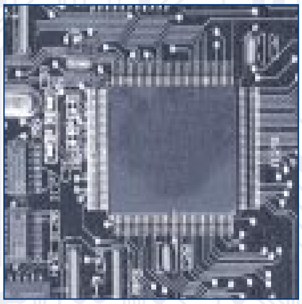
A TDR instrument specifically designed for the measurement of PCB controlled impedance will be able to:

- **Be operated in a manufacturing environment by non technical operators**
- **Produce graphs of Impedance versus Distance over the length of the test coupon**
- **Datalog results and produce reports suitable for presentation to the final customer**
- **Store test files which contain all of the specifications for each type of coupon and automatically set up the TDR**

Polar Instruments designs and manufactures Controlled Impedance Test Systems that are specifically designed for measuring impedances on PCB's and has become a world leader supplying instruments to premier PCB companies throughout the world.



International Standards



There are a number of International Standards that relate to controlled impedance PCB's . At the time of this booklet (1996), two in particular are:

IPC-D-317-A, Design Guidelines for Electronic Packaging Utilizing High-Speed Techniques

Published by:

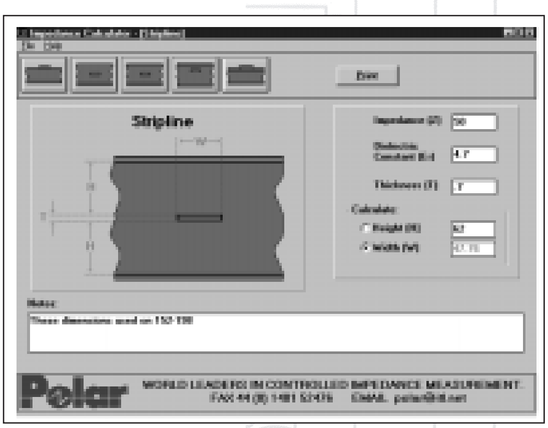
Institute for Interconnection and Packaging Electronic Circuits
2215 Sanders Road, Northbrook, Illinois 60062 - 6135, USA
Fax: (847) 509 9798

Controlled Impedance for Printed Circuit Boards (Draft Version), Addition to IEC 326-3: Reference Number 52 (Secretariat).U, date for additions to be included into IEC 326-3 to be decided later

Draft published by:

Printed Circuit Interconnection Federation
45 Beaufort Court, Admirals Way, South Quay, London E14 9XL, UK
Fax: 0171 515 1188

Impedance Equations



The above Standards quote equations that may be used to calculate nominal impedances of different configurations. However Polar Instruments has a Field Solving Controlled Impedance Calculator which provides a more advanced approach to impedance calculation.

Single Ended vs Differential Impedances

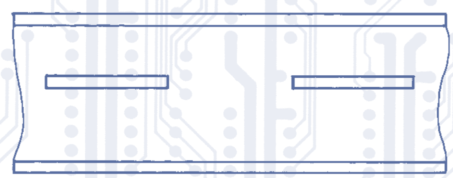
All references made so far to controlled impedances have assumed a single conductor trace and a return path (ground plane). This is known as a single ended configuration.

Certain PCB's also use Differential Impedances where there are **two matched traces** as well as the ground return. Three common configurations are shown.

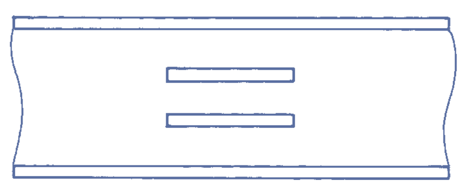
In addition to all of the previously mentioned factors that affect impedance (trace width, trace thickness, laminate height and laminate material) the **spacing between the two traces will also affect the value of differential impedance.** In use, the electronic component on the PCB will drive the two traces with equal magnitude but opposite polarity signals. The closer the spacing between the two traces, the greater the effect they will have on each other via electrical coupling and this will reduce the value of differential impedance.

Typical differential impedances are around 100 ohms. They consist of 2 matched traces that individually have dimensions which would give a single ended impedance of just over 50 ohms. The coupling of the two traces, each just over 50 ohms results in a differential impedance of around 100 ohms.

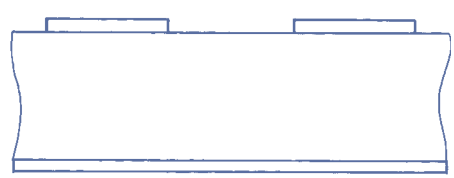
Differential test coupons are made in the same way as for single traces but the appropriate test coupon layer will contain two traces identical to those on the main PCB with exactly the same spacing between them as the main PCB. Testing is done using a TDR with differential capability which will have two test cables which are connected to the test coupon to make the differential measurement.



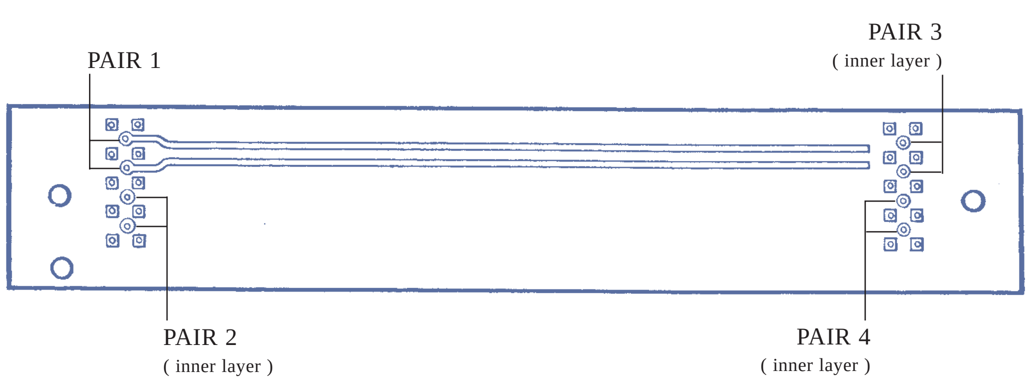
EDGE COUPLED STRIPLINE



BROADSIDE COUPLED



EDGE COUPLED MICROSTRIP



1. Traces may be edge coupled (on same layer) or broadside coupled (on adjacent layers)
2. In above design, test traces are connected to one side only, allowing 4 pairs of differential traces to be measured (2 on left side, 2 on right side)

CITS (Controlled Impedance Test System)

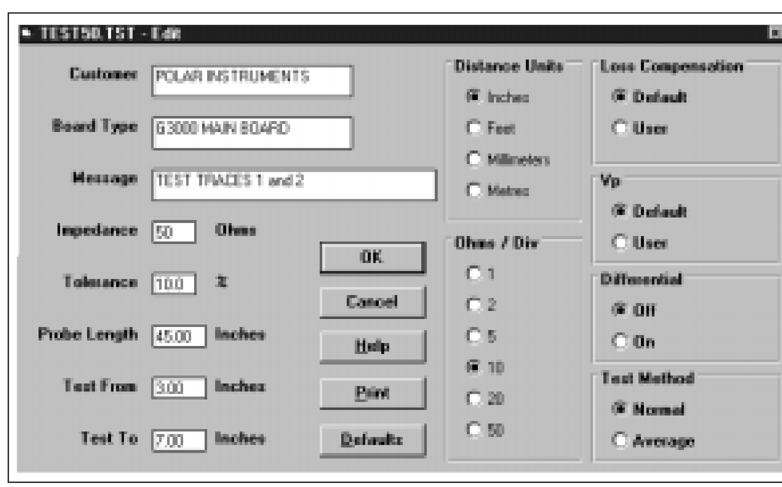
Polar Instruments manufactures a TDR instrument which is specifically for the measurement of controlled impedances on PCB's. The system comprises of an instrument containing the TDR which connects to an industry standard PC (Personal Computer), running special software that supports CITS.

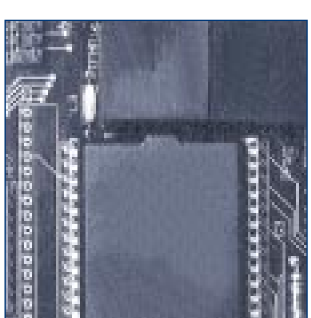
The software has features that are designed to operate within a PCB manufacturing environment and is able to:

- **Be used by non technical operators**
- **Produce graphs of Impedance versus Distance over the length of the test coupon**
- **Datalog results and produce reports suitable for presentation to the final customer**
- **Store test files which contain all of the specifications for each type of coupon and automatically set up the TDR**

Additionally CITS is provided with special handheld probes that make reliable and consistent contact with the test coupon.

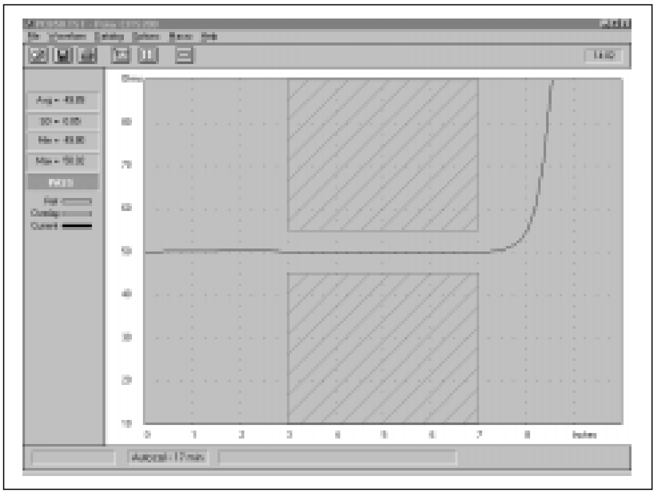
You can set up a Test File which contains all of the relevant data for a particular coupon as shown in the screen below. The computer will save this Test File so that for each batch of boards, all of the parameters are automatically set up.



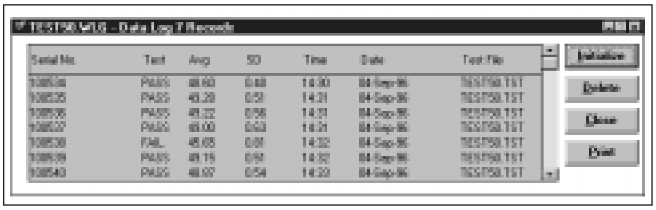


CITS

When a Test File has been selected, the operator places the probe on the correct pads on the test coupon and presses a footswitch to make the measurement. CITS then automatically generates a graph of Impedance vs distance as shown and indicates a PASS or FAIL.

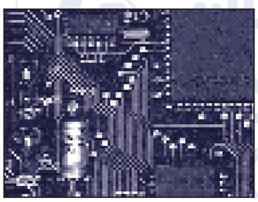


CITS includes an automatic datalog option allowing all the data to be logged against a coupon's serial number as shown. This is available as a printout. The data can also be imported into a spread sheet to generate SPC (Statistical Process Control) information



Polar Instruments

CITS has become accepted as an industry standard for the measurement of controlled impedances in a manufacturing process. Polar Instruments Ltd, approved to International Quality Standard ISO9001, is committed to continuous development to ensure that our products reflect customer requirements. Please contact us or our distributor if you would like further information or have any comments concerning this booklet.



For USA / Canada, please contact

Polar Instruments, Inc
5373 Guide Meridian
Building D
Bellingham
WA 98226
USA

Tel/fax: (360) 398 1946 Customer tel: (800) 328 0817

Head Office

Polar Instruments Ltd
Garenne Park, St. Sampson, Guernsey, Channel Islands GY2 4AF, UK.
http://www.polar.co.uk

Tel: +44 (0)1481 53081 Fax: +44 (0)1481 52476 Email: polar@itl.net

$9.95

LIT145