Circuits with controlled impedance

1. Introduction

Owing to significantly increasing transmission frequencies, i.e. shorter pulse rise times of electronic components, in high frequency technology, it has become necessary to treat the conductors concerned as a component. Depending upon a number of parameters, HF signals are reflected on circuit boards. This means that the characteristic impedance differs from the output impedance of the transmitting component. The proper transmission of the signal is thus no longer guaranteed.

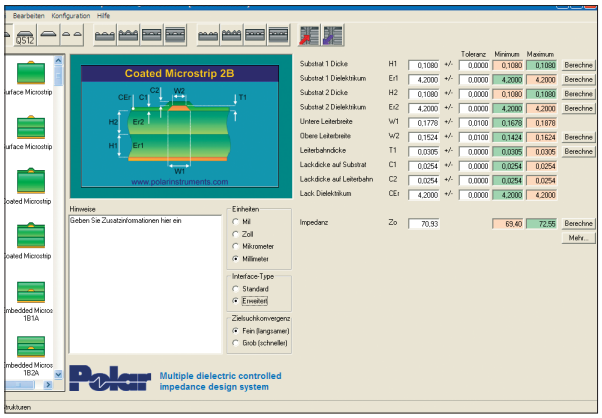
contag has created the option of checking the required impedance on the customer's circuit board, so as to modify the circuit board, or the tracks and the layer structure if necessary. The impedance is largely determined by the track geometry, the structure of the layers and the dielectric constant (ε_r) of the materials used.

After making the circuit board, the impedances are checked and recorded. The results of the measurements are available on request at any time.

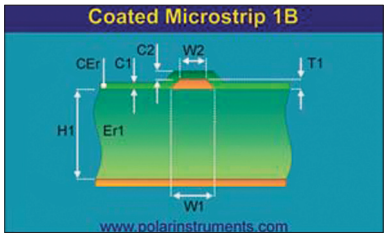
2. Impedances on the circuit board

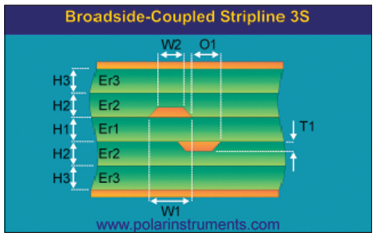
The impedance specified by the customer will be checked during the PCB maker's preparatory work to see if it can be made. Depending on the definition of the layers in the stack and on the layout itself, from a set of 93 different models, the correct one must be selected to be able to calculate the impedance. As a result of this, a layer structure will be produced together with any necessary modifications to the geometry of the relevant tracks.

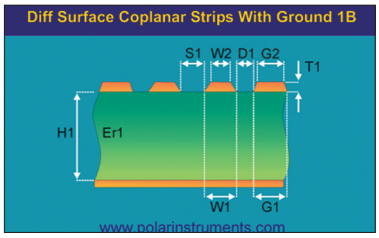
The standard tolerance used for an impedance is ±10%. Depending upon various PCB characteristics, on request an improved tolerance (down to ±5%) can be offered.



3. Typical models for calculations







To be able to actually reach the calculated impedances in the printed circuit board, the manufacturer must understand their production processes very well and have good control of them. The angle of the track edges (resulting from W1, W2 and T1), the pressed thickness depending on the copper coating (H), the thickness of the solder resist varnish (C1, C2) and more, all influence the impedances on the circuit board. The tolerance range of all processes have to be considered here. These values are then added to the formulae for the calculation.

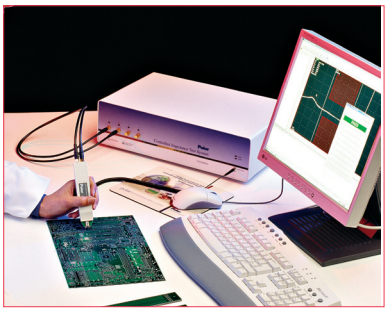
Depending on the complexity of the circuit board, some of the impedances of the circuit board can be checked during the manufacturing process. For this, new models for the corresponding situation (actual state of the circuit board) are separately defined and the results determined.

4. Dielectrics

To make controlled impedance circuit boards, differing materials are used. FR4 is standard here too. However, in applications with very high frequencies (>1 GHz), requirements such as low dielectric loss or a lower ε_r are of increasing importance.

Here, contag can offer a large portfolio of usable materials. Many of these are seen as standard and are regularly used, others are available at short notice. You will find further information in the "Materials".

5. Quality control



As impedances can no longer be measured once the components have been fitted, it is necessary to check and confirm the values prior to assembly. When the job is prepared, so-called "test coupons", which are similar to the relevant tracks, are added to the net area of the production blank. The impedance of the test coupons represent those of the actual circuit board. CONTAG has a CITS900s in their final inspection, the newest test equipment made by POLAR.

6. Design notes

Depositing electroplated copper results in a relatively imprecise and uneven thickness compared to copper foil (for internal layers). For this reason, it is important to think about the manufacture while a circuit board is being developed. In many cases it can make sense to manufacture the layout using tracks with a defined impedance and the associated earth surface on a core, as an internal layer. In this way one of the most important factors, which can falsify the resulting impedance, is eliminated.

The conductors for differential signals should not be too narrow (< 150 µm) and should be routed with sufficiently large separation. To control the impedance of a layout, the circuit board manufacturer has to optimise the layout for their production processes.

Talk to our contag team (+49 30 351 788 – -0 or team@contag.de). We will look together for a functional, optimised and cost-effective solution.