The Controlled Impedance Measurement on the PCB

[1] Min Ju Park, [1] Jae Kyung Lee, [1] Dal Hwan Yoon, [2] Seung Gi Min
[1] Dept. of Electronics Engineering, Semyung University
[2] Human Medtek., Co., Ltd.
[1] San21-1, Shinwol-dong, Chechon, Korea
Tel. +82-43-649-1790, Fax. +82-43-644-6165
[1] E-mail : yoondh@semyung.ac.kr

Abstracts: The digital systems include the noise in power supply, ground and packaging due to a simultaneous switching of signal, signal reflections and distortions on single and multiple transmission lines. The requirement for the controlled impedance on a PCB can be both a critical success factor and a design challenge. So, the invented tool simulates the tracks controlled impedance with the test coupon. It can saves the design time and supports the economical PCB design.

Keyword: impedance, PCB design, control, test coupon.

I. Introduction

The performance of a communication system will be affected on the length of a signal transmission line, the loss rate of the materials, line types and an impedance discrepancy.

Recently, most of digital systems employ the frequency band of various layer, the scale of a system is going to the small size and be light in weights. Also, the PCB style of corresponding to such a tendency being changed a simple layer multiple layers[1].

On design the signal transmission path, we must consider carefully the reflection signal and the leakage signal due to the electromagnetic interference between neighbor lines. The signal loss factor in terms of the materials, the dielectric and slain effect be considered to include the precision of an analysis[2,3].

In general, the system manufacturers have thought that the performance of a system depend on a superior materials, and then always the rate of the cost to the performance. But the fidelity lacks of a transmission signal have been the performance fall off[4-6].

In this paper, we have developed an impedance measure tools that can design easily the PCB at a materials of the transmission line, a dielectric and

the pattern of the line. Also, we explain the theoretic of tracks calculation in terms of a dielectric and a line patterns and shows the experimental result.

II. Theory of a controlled impedance

On the PCB(printed circuit boards), the line tracks must be made by a controlled impedance. Wudell[2] have proved the equation of estimating an impedance. The equation is the function that be made by the geometrical line between a signal path and a reference plane, the distance of dielectrics and the dielectric constant of discriminating layers.

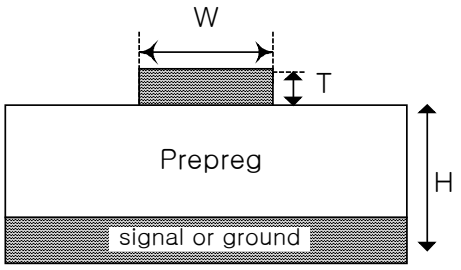
The circuit on a PCB can be affected on the formats of a circuits, the width(W) and the thickness(T) of

transmission path, a dielectric constant(ε_r). When we design the PCB, important parameters are a via hole, the pad and connected angle, a scratch, a one-sided declination between layers and the bending degree of a PCB.

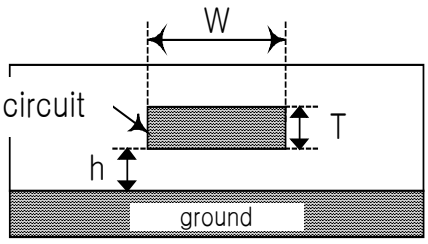
The PCB manufactures will carefully manage the width of a circuit and the allowable tolerance of an insulated thickness. The PCB designers have to follow an allowable tolerance and acknowledge the characteristics of materials and a making process.

Polar[8] have proposed twelve types of impedances model. Four types of there are showed in figure 1. In figure1, (a) the type of a surface microstrip has design the circuit on the prepreg of the PCB. Where W is the track width, T is the track thickness and H is the height of a prepreg. (b) the type of an embedded microstrip is that the circuit is located in the middle of PCB, where h is the distance between track and ground. (c) the stripline type have the circuit between a source signal and a ground plane. (d) the dual stripline type have two circuits between a source signal and a ground plane and have differential distances of a B and a C between any standard planes. Thus, many formats of a PCB are made by the multi-layer.

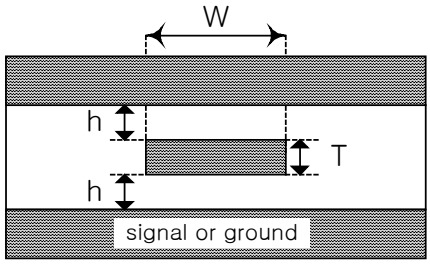
IPC-2141[3] have proposed another estimation method of an impedance, there are some differences in terms of a design formats.



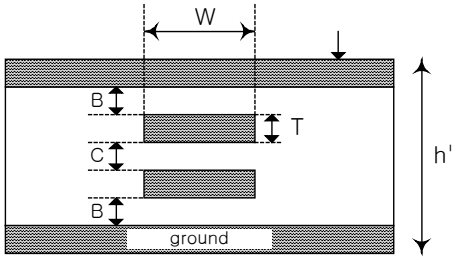
(a) A surface microstip formats



(b) An embedded microstip formats



(c)A symmetrical stripline formats



(d) The dual stripline formats

Fig 1. PCB formats to control an impedance.

Each controlled impedance equation of several PCB types in fig.1 is as follows, the impedance of the surface microstip format is

$$Z_0 = \frac{87.0}{\sqrt{\varepsilon_r + 1.41}} \ln\left(\frac{5.98H}{0.8W + T}\right) \qquad (1)$$

An embedded microstip format is

$$Z_0 = \frac{K}{\sqrt{0.805\varepsilon_r + 2}} \ln\left(\frac{5.98h}{0.8W + T}\right) \qquad (2)$$

The symmetrical stipline format is

$$Z_0 = \frac{60}{\sqrt{\varepsilon_r}} \ln\left(\frac{4(2h+T)}{2.1(0.8W+T)}\right) \quad (3)$$

The dual stripline format is

$$Z_0 = 80\left[\frac{1-\frac{B}{4(B+C+T)}}{\sqrt{\varepsilon_r}}\right] \ln\left(\frac{1.9(2B+T)}{0.8W+T}\right) \quad (4)$$

In these days, the digital system designated by the PCB has very high speed. Especially, the DSP due to such a high speed has faced with an electromagnetic wave delay, the reflection of the transmission signal, a signal loss and the interconnection complexity due to high connection density. In order to solve these problems, we are to control the circuit width, the dielectric constants, the permeability and the thickness of the PCB. These are an important parameter for a manufacturer. Because the transmission efficiency of the circuit depend on its properity.

An experimental coupon must be made of easily as possible as an impedance measurement. In general, a test coupon size is 200mm×300mm, it must has the same track structure and the layer as an actual manufactured goods.

The working panel for a test can obtain a high reliability by using two coupons, concurrently. Actually, many manufacturers have designated the PCB in an experienced method without having an experimental impedance measurement. Thus, the PCB manufacturer must produce the coupon by controlling the circuit width and an interval of insulated layers, the repeated experiments and measurements after an impedance calculation be concurrently handled. Then we can make out the difference table between the calculated results and the experimental results, and must grasp the variability between the theoretic value and the manufacturer's making-process.

Ⅲ. The measurement tool design

We need the followed information to design the PCB with satisfying the customer's demand specification. What ohm's impedance circuit is needed? How much is the total thickness and the layer? Shall we insert the signal line in any layer? And finally, How much is the width of the circuit?

First of all, the manufacturer must grasp the characteristics and the uses of a system, and must demand the specification. And then we must find out an impedance value and allowable tolerance, layers, the width of a circuit, an insulated interval of a layer.

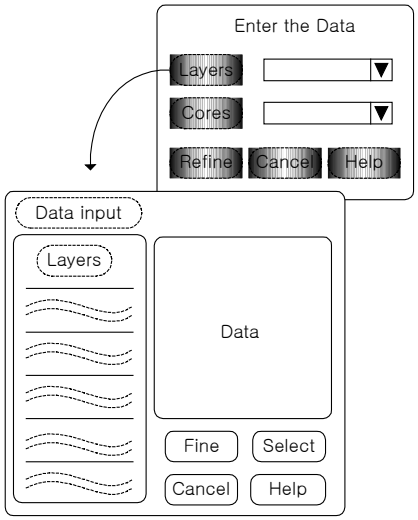


Fig. 2 The working monitor

Finally, we produce the PCB of conserving the

careful terms. Figure 2 shows the measurement tool for monitoring these specifications. First, the whole monitor is displayed by entering the user name and password.

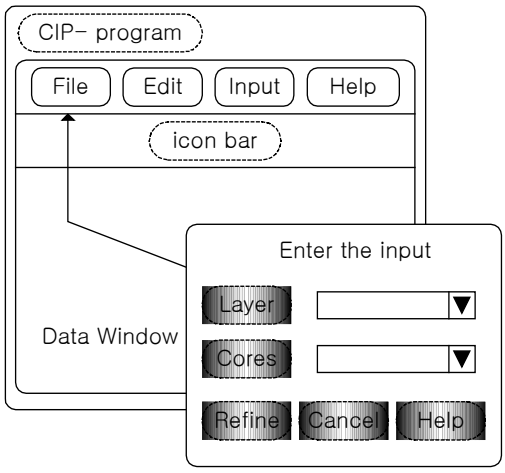


Fig. 3 The data input window

If we click the file menu, the layers number of the PCB and the signal path number are displayed. The interactive activity create the new file and the required database.

Figure 3 shows the data input window after it has entered the numbers of layers and cores. An input data call the existed database or can try to enter new data. In case of using frequently the specification, we don't need to make a new data.

Figure 4 shows an impedance variability rate to the track width of the surface micro stripline.

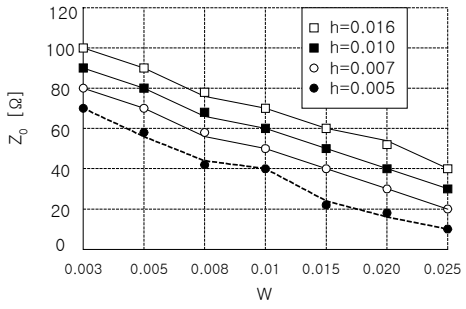


Fig. 4 An impedance variability rate

The horizontal line is the width, and the vertical line is an impedance. Figure 5 shows the change rate of an impedance to the s/h.

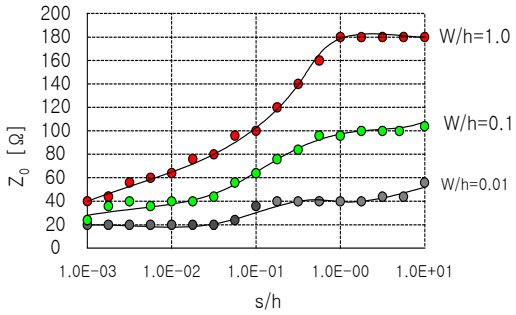


Fig. 5 An impedance to the s/h.

Figure 6 shows the change rate of an impedance to the W/h.

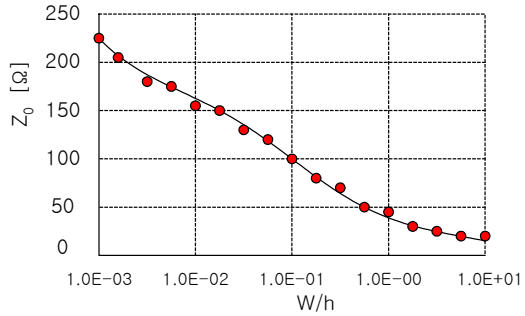


Fig. 6 An impedance to the W/h.

Ⅳ. Conclusion

Recently, the effort of obtaining the high signal fidelity has been worked in various areas. The complexity and function a system using GHz have required for a high circuit concentration, and then an effective PCB design.

Especially, we always have the issue about the impedance in middle of the transmission path pattern, and then it has become the important factor to

determine the error rate of the signal. Also, it is an important factor of minimizing an electromagnetic noise. When we manufacture the PCB with a design specification, the process is very complex. In order to design easily the PCB, an automatic impedance measurement tool is necessary. In this paper, we have developed measurement tool, and it could have reduced the manufacturing process using the predetermined specification. And then we precisely can design the PCB and can conserve the signal fidelity.

Conference

[1] www.polarinstruments.com

[2] Polar inc., “Calculation of Controlled Impedance, 2D Field Solving in the SI6000 and CITS25”, Application Notes 135,

[3] F. Y. Chang, “Transient Analysis of Lossless Coupled Transmission Lines in Nonhomogeneous Dielectric Medium,” IEEE Trans. Microwave Tech., pp. 616-626, Sep., 1970

[4] H. W. Johnson and Martin Graham, “High-speed Digital Design : A Handbook of Black Magic,” PRT Prentice-Hall, 1993

[5] M. I. Montrose, “EMC and the Printed Circuit Board,” IEEE Inc., pp. 175-182, 1996

[6] Werner John, “Design of Printed Circuit Boards as a Part of an EMC-adequate System Development,” IEICE Trans. Comm. vol. E80-Bm No. 11, Nov. 1997,

[7] Hart, Bryan, "Digital Signal Transmission," Pub : Chapman and Hall 1988

[8] Harrington, Roger F, "Field Computation by Moment Methods," Pub : MacMillan, 1968

[9] CITS25, "Differential Controlled Impedance Calaulator," Polar Instruments Ltd, http://www.polars.co.uk, 1998

[10] Sadiku, Matthew No. “Numerical Technigues in Electromagnetics,” Pub : CRC Press 1992

[11] Silvester P. P, “Microwave Properties of Microstrip Transmission Lines, IEE Proc. of Vol. 115, No. 1, pp 43-48, Jan. 1969