

What is impedance control PCB?

Impedance Control PCB is the characteristic impedance of a transmission line formed by PCB conductors. It is relevant when [high-frequency](#) signals propagate on the PCB transmission lines. PCB Impedance Control is important for signal integrity: it is the propagation of signals without distortion.



 QUOTE





The impedance of the circuit board is determined by the physical dimensions (line wide/space) and materials of the circuit and is measured in Ohms (Ω).


PCB impedance control has been one of the essential concerns and challenging problems in high-speed [PCB design](#). As an impedance control designer, you should know what affect PCB impedance and how to calculate impedance.

[Request Impedance Control PCB Quote](#)

1.The following parameters largely determine the impedance of a PCB:

- 1) Distance of signal layer and potential
- 2) Conductor geometry
- 3) Trackwidth
- 4) [Copper thickness](#)
- 5) Permittivity ϵ_r

2.Impedance and Delay Calculation Formula of Transmission Lines on PCB

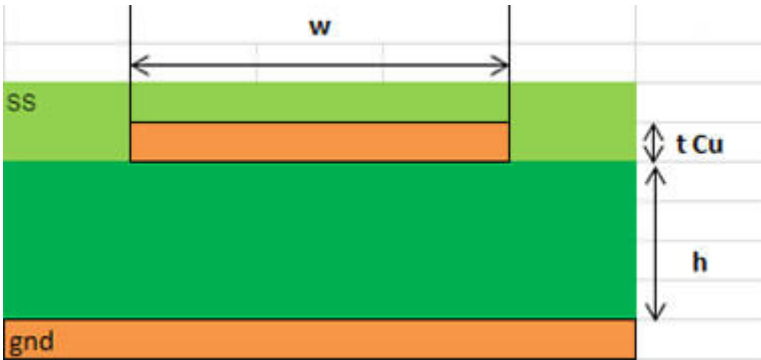
Transmission Lines on PCB	Impedance and Delay Calculation Formula
 <p>Figure 2 Microstrip Transmission Lines</p>	$Z_0 = \frac{87}{\sqrt{\epsilon_r + 1.41}} \ln\left(\frac{5.98H}{0.8W + T}\right) \text{ Ohms}$ <p>(Valid when $0.1 < W/H < 2.0$ and $1 < \epsilon_r < 15$)</p> $t_{PD}(\text{microstrip}) = 85\sqrt{0.47\epsilon_r + 0.67}$
 <p>Figure 3 Symmetrical Stripline Transmission Lines</p>	$Z_0 = \frac{60}{\sqrt{\epsilon_r}} \ln\left(\frac{4H}{0.67\pi(T + 0.8W)}\right) \text{ Ohms}$ <p>(Valid when $W/H < 0.35$ and $T/H < 0.25$)</p> $t_{PD}(\text{stripline}) = 85\sqrt{\epsilon_r}$



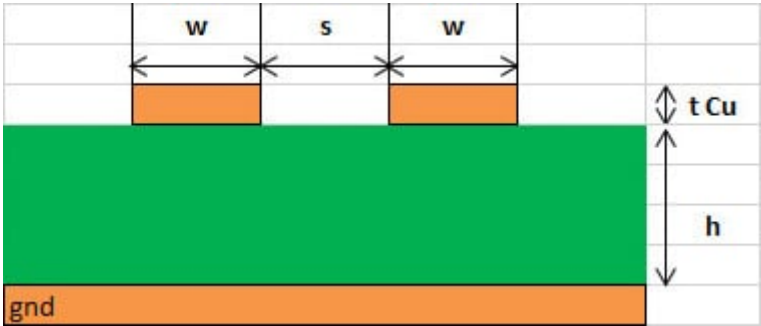
 [QUOTE](#)

Formula Restrictions: $0.1 < w/h < 3.0$

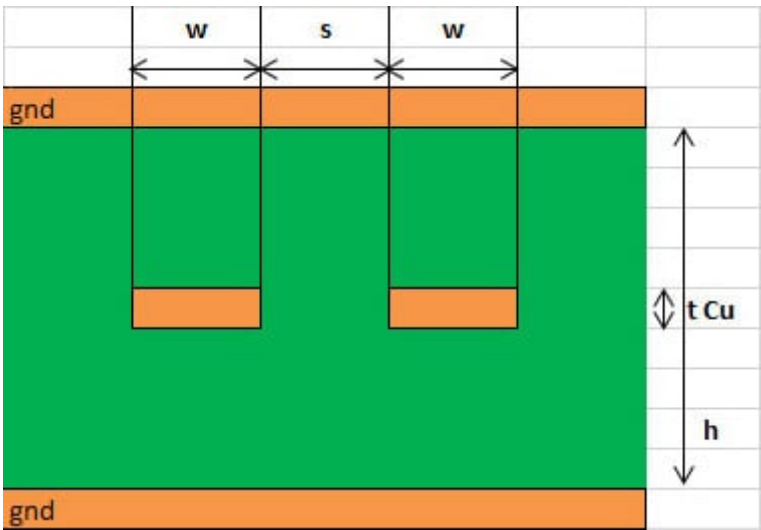




Formula Restrictions: $0.1 < w/h < 3.0$



Formula Restrictions: $0.1 < w/h < 3.0$



Formula Restrictions: $0.1 < s/h < 3.0$

Here is some [free PCB software](#) that provides impedance control calculations and online calculators as a reference.

Selecting Foil-Built or Core-Built PCBs to Support Impedance Control

RayMing Technology normally suggests impedance designers choose Foil-Built PCBs in order to make the most economical PCB. That being said, our [Prototype PCB Assembly](#) is very flexible, and we can use Core-Built PCBs. Foil-Built PCBs not only tend to be more economical than Core-Built PCBs, but they are also slightly easier to fabricate. The pictures below explain the differences between the Foil-Built and Core-Built PCBs.

Foil Build						
	SOLDERMASK			0.5		
L1	FOIL		H oz	0.6 + 1.4	Plating	
	Prepreg	2x1080		5.5		
L2			1 oz	1.2		
	Core	5		5		
L3			1 oz	1.2		
	Prepreg	2x1080		5.5		
L4	FOIL		H oz	0.6 + 1.4	Plating	
	SOLDERMASK			0.5		
TOTAL BOARD THICKNESS CALCULATED				23.4		
TOTAL BOARD THICKNESS REQUIRED				23 +/- 10%		

Core Build						
	Core	5.5		5.5		
L4	FOIL		H oz	0.6 + 1.4	Plating	
	SOLDERMASK			0.5		
TOTAL BOARD THICKNESS CALCULATED				23.9		
TOTAL BOARD THICKNESS REQUIRED				23 +/- 10%		



 [QUOTE](#)

	Core	5.5		5.5			
L4	FOIL		H oz	0.6	+	1.4	Plating
	SOLDERMASK			0.5			
TOTAL BOARD THICKNESS CALCULATED				23.9			
TOTAL BOARD THICKNESS REQUIRED				23 +/- 10%			



Foil-Built PCB board uses one less core than the Core-Built PCBs in the stack-up. The outside consists of aluminum foil. In addition, foil in different copper weights is much easier to purchase. Since Foil-Built PCBs are made of aluminum, they can be used on any builds, regardless of the primary laminate material used. PP (Pre-pregs) are also [less costly](#) than cores, especially if they are 5 mil or thinner.

Core-Built PCB board has cores on the outside, so there is no need to use aluminum foil. Depending on material availability, it may be difficult to acquire a core with uneven copper weights. This forces the [PCB manufacturer](#) to etch down the cores, which is costly since a good deal of labor is involved. Except for labor, the PCB factory must use a higher copper weight than what appears on the board, increasing material cost.

Nowadays, PCB designs and [electronic components](#) become smaller, faster – in other words, more complicated. PCB Impedance Control is becoming more critical. RayMing has ten years of PCB manufacturing experience, and we can provide full support when designing a PCB impedance board.

PCB Impedance Design and calculator

1. Main types and affecting factors of PCB impedance design

The opposition encountered during current transmission in a DC circuit is called resistance, and the opposition encountered by the current in an AC circuit is called impedance. The opposition encountered by the transmission signal in the high frequency (>400MHZ) circuit is called the characteristic impedance. In high frequency, the transmission signal copper wire on the PCB can be regarded as a conductive line composed of a series of equivalent resistance and a parallel inductance. This equivalent resistance is as small as it can be ignored. Therefore, when we analyze the signal transmission of a PCB at high frequency, we only need to consider the effects of the stray distribution of series inductance and parallel capacitance. We can get the following formula:

$Z_0 = R + \sqrt{L/C} \approx \sqrt{L/C}$ (Z_0 is the characteristic impedance value)

- When the digital signal is transmitted on the board, the characteristic impedance of the PCB trace must match the electronic impedance of the head and tail components. If it does not match, the transmitted signal energy will be reflected, attenuated, delayed, or lost, resulting in miscellaneous messages.
- The higher the electronic impedance of the electronic component, the faster its transmission rate, so the characteristic impedance value of the circuit board must be increased accordingly to match it.
- In addition to Z_0 , the PCB used for [radio frequency](#) communication sometimes emphasizes the board itself has a low ϵ_r (dielectric constant) value and a low D_f (dielectric loss factor) value. The transmission speed of high-frequency signals in the medium is C/ϵ_r . It can be seen that the smaller the ϵ_r , the faster the transmission speed, which is why high-frequency materials with a low dielectric constant are used. D_f affects the distortion of the signal in the medium transmission process. The



QUOTE

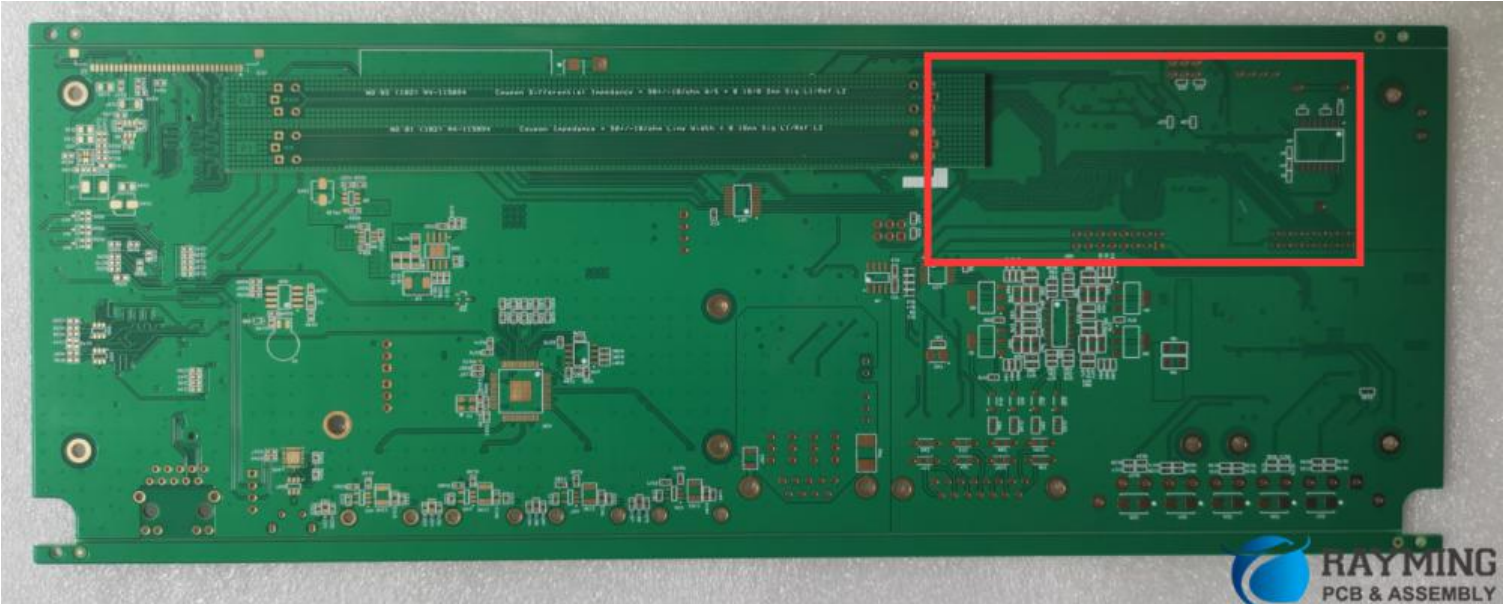
promotes the integration of integrated circuits towards high density, small volume, and single parts. These are the reasons that lead to a high frequency of future [PCB materials](#). The use of high-speed digital circuits means that the impedance of the circuit must be controlled, low distortion, low interference, low crosstalk, and elimination of



electromagnetic interference (EMI). Impedance design is gradually crucial in PCB design. In PCBs with high-frequency signal transmission, the control of characteristic impedance is critical, and characteristic impedance is the core of solving signal integrity problems. At the front-end of PCB manufacturing, the front part of the system is responsible for the simulation calculation of impedance and the design of the impedance bar. Customers have become more stringent in impedance control. The number of impedance controls has increased how to quickly and accurately design impedance is a problem that greatly concerns PCB designers.

[Request Impedance Control PCB Quote](#)

2.Main types and affecting factors of impedance



Impedance(Z_0) definition: The total opposition to an alternating current flowing through it at a known frequency is called impedance(Z_0). For PCBs, it refers to the total opposite of a certain circuit layer (signal layer) to its closest reference plane under high-frequency signals.

- The types of impedance
- Characteristic impedance

In electronic information products such as computers and wireless communications, the energy transmitted in the PCB circuit is a square wave signal composed of voltage and time. The opposition it encounters is called characteristic impedance.

- Differential Impedance

Two identical signal waveforms with opposite polarities are input at the driving end, respectively transmitted by two differential lines, and the two differential signals are subtracted at the receiving end. Differential impedance is the impedance between two lines.

- Odd-Mode Impedance

The impedance of one line to the ground is the same as the second line.

- Even-mode impedance

Impedance when two same signal waveforms with the same polarity are input at the drive end and the two wires are connected



 [QUOTE](#)

two lines, it is usually greater than the odd mode impedance.

Among them, characteristic impedance and differential impedance are common impedance, and common-mode impedance and odd-mode impedance are rare.



- Analysis of the factors affecting impedance from the perspective of PCB manufacturing

W — Line width/line space: The line width increases, the impedance decreases, and the distance increases, the impedance increases.

H — Insulation thickness: the thickness increases, and the impedance increases.

T — Copper thickness: the copper thickness increases, and the impedance decreases.

H1 — Solder mask thickness: the thickness increases, and the impedance decreases.

Er — The dielectric constant: the DK value increases, and the impedance decreases.

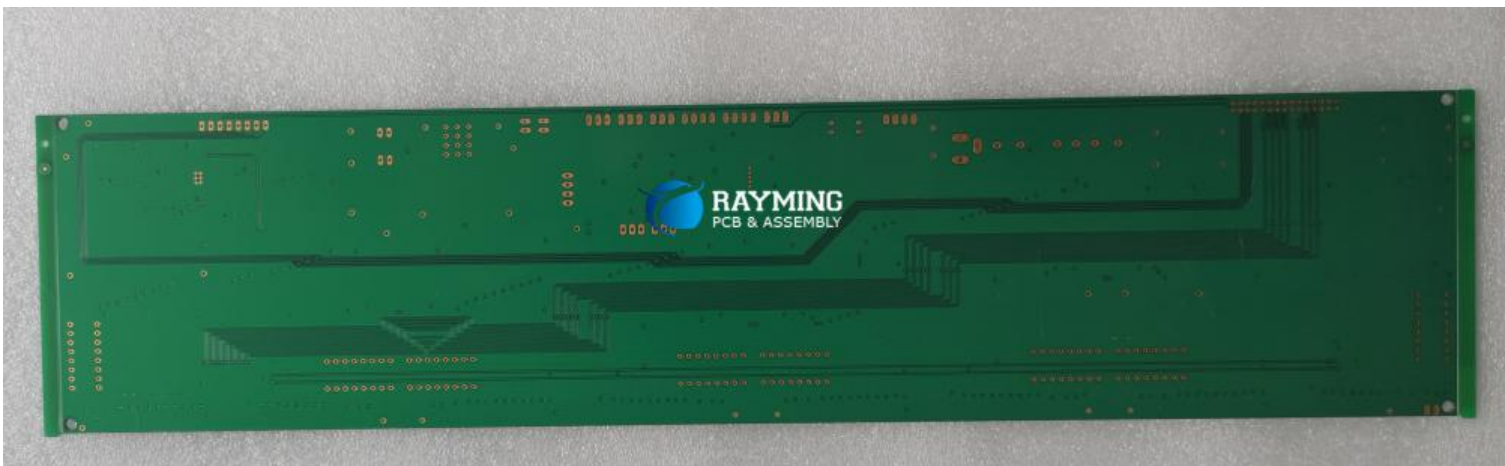
Undercut — W1-W: the undercut increases, and the impedance increases.

In addition, when the surface process is [immersion-gold](#) (or gold plating), the etching process of the outer layer is different from other [surface processes](#), and the line compensation is different. The impedance calculation result of the former will be 3-5 ohms larger, so the resistance value of the gold plating process and other process switching needs to be adjusted.

3.Impedance calculation automation

The most commonly used impedance calculation tool in our industry is the Si8000 Field Solver provided by Polar. Si8000 is a brand-new boundary element method field-effect calculator software. It is based on the easy-to-use user interface of the early Polar impedance design system we are familiar with. This software contains various impedance modules, and personnel can calculate impedance results by selecting specific modules, inputting related data such as line width, line spacing, inter-layer thickness, copper thickness, and Er value. A PCB impedance control number can be as few as 4 to 5 groups or as many as dozens of groups. The control line width, inter-layer thickness, copper thickness, etc., of each group are also different. If you check the data one by one and then manually input the relevant parameters and then calculate, it is time-consuming and error-prone.

To ensure the quality of signal transmission, reduce EMI interference, and pass relevant impedance testing and certification, it is necessary to conduct an impedance matching design for PCB key signals. Considering common calculation parameters, TV product signal characteristics, [PCB Layout](#) actual needs, SI8000 software calculations, PCB supplier feedback information, etc., finally come to this recommended design. It is suitable for most PCB suppliers' process standards and PCB board design with impedance control requirements.



- Ground package design: line width/spacing 7/5/7mil ground wire width ≥ 20 mil signal and ground wire distance 6mil, add ground vias every 400mil.



- Design without ground package: line width/spacing 10/5/10mil, the distance between the differential pair and the pairs ≥ 20 mil (not less than 10mil in special cases). It is recommended that the entire group of differential signal lines is shielded with grounding, and the distance between the differential signal and the shielding ground ≥ 35 mil (not less than 20mil in special cases).

90 ohm differential impedance recommended design

- Ground package design: line width, line spacing 10/5/10mil ground wire width ≥ 20 mil signal and ground wire distance 6mil or 5mil, add ground vias every 400mil
- Design without ground package: line width, line spacing 16/5/16mil, the distance between the differential pair and the pairs ≥ 20 mil, it is recommended that the entire group of differential signal lines should be shielded with ground, and the distance between the differential signal and the shielding ground ≥ 35 mil (In special cases, it cannot be less than 20mil).

Note: Prioritize the use of package ground design. However, if the line is short and there is a complete ground plane, it can be designed without package ground.

Calculation parameters:

FR-4, thickness 1.6mm+/-10%, dielectric constant 4.4+/-0.2, copper thickness 1.0 oz (1.4mil), solder mask thickness 0.6±0.2mil, dielectric constant 3.5+/-0.3.

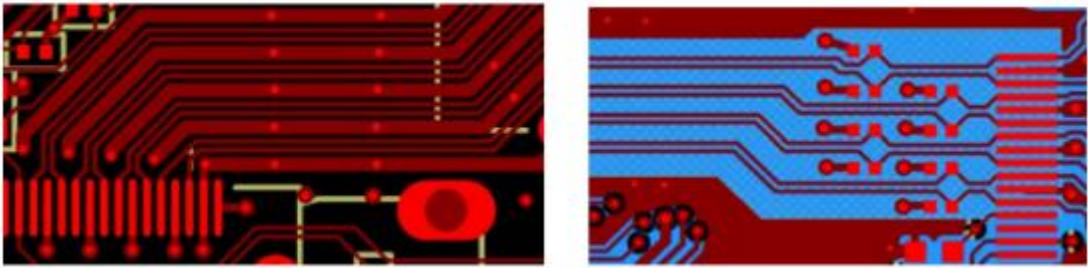


Fig.1 Design of ground package Fig.2 Design without ground package

2) Four Layer impedance design

100 ohm differential impedance recommended design

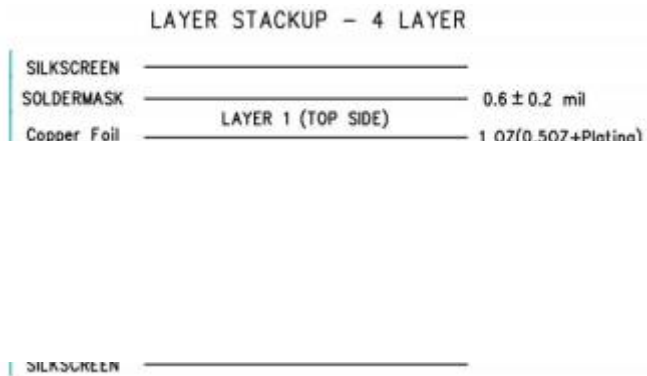
line width, spacing 5/7/5mil differential pair and the distance between the pair ≥ 14 mil (3W criterion);

Note: It is recommended that the entire group of differential signal lines is shielded with ground, and the distance between the differential signal and the shielded ground line is ≥ 35 mil (in special cases, it cannot be less than 20mil).

90 ohm differential impedance recommended design

line width, spacing 6/6/6mil differential pair, and the distance between the pair ≥ 12 mil (3W criterion).

Note: In the case of a long differential pair trace, it is recommended that the USB differential line wrap the ground at a distance of 6 mils on both sides to reduce the risk of EMI (with and without ground, the line width and line spacing standards are consistent).



QUOTE

Calculation parameters:



FR-4, thickness 1.6mm+/-10%, dielectric constant 4.4+/-0.2, copper thickness 1.0 oz (1.4mil), copper clad substrate(PP) 2116 (4.0-5.0mil), dielectric constant 4.3+/-0.2, solder mask thickness 0.6±0.2mil, dielectric constant 3.5+/-0.3.

Stack-up:

Silkscreen
Solder mask
Copper
Prepreg
Base material
Prepreg
Copper
Solder mask
Silkscreen

3) Six Layer impedance design

The impedance design of the outer trace is the same as that of the four layer board.

The inner trace is generally more plane layer than the surface trace, and the electromagnetic environment is different from the surface.

The following is the third layer trace impedance control recommendation.

100 ohm differential impedance recommended design

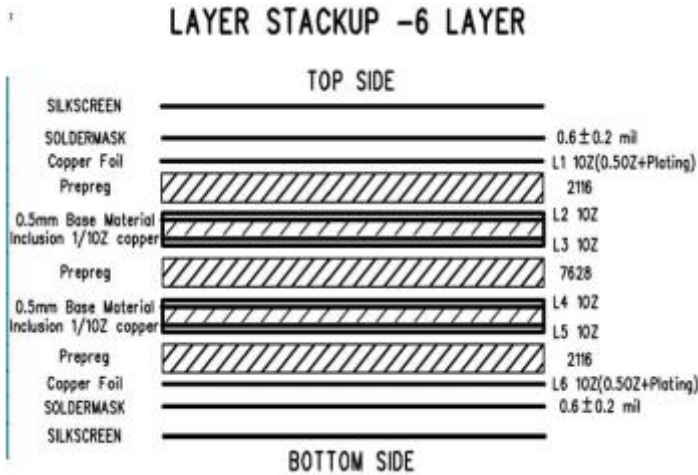
line width, spacing 6/10/6mil.

The distance between the differential pair is ≥20mil (3W criterion).

90 ohm differential impedance recommended design

line width,line spacing 8/10/8mil.

The distance between the differential pair is ≥20mil (3W criterion).



Calculation parameters:

FR-4, thickness 1.6mm+/-10%, dielectric constant 4.4+/-0.2, copper thickness 1.0 oz (1.4mil), copper clad substrate(PP) 2116 (4.0-5.0mil), dielectric constant 4.3+/-0.2, solder mask thickness 0.6±0.2mil, dielectric constant 3.5+/-0.3.

Stack-up:



Silkscreen
Solder mask
Copper
Prepreg
Base material
Prepreg
Base material
Prepreg
Copper
Solder mask
Silkscreen

2. For layers more than six layers, please design by yourself according to relevant rules or consult relevant personnel to determine the stack-up and trace
3. If there are other impedance control requirements with special circumstances, please calculate by yourself or consult relevant personnel for alternative design solutions.

Note:

- Many situations affect impedance, and PCBs that require impedance control still need to indicate impedance control requirements in the PCB design data.
- 100 ohm differential impedance is mainly used for HDMI and LVDS signals, among which HDMI must pass relevant certification.
- 90 ohm differential impedance is mainly used for USB signals.
- Single-ended 50 ohm impedance is mainly used for some DDR signals. Given the large DDR particles, some adopt an internally adjusted matching impedance design. The design is based on the demo board provided by the solution company. This design guide is not recommended;
- Single-ended 75 ohm impedance is mainly used for analog video input and output. There is a 75 ohm resistor in the circuit design to match the ground resistance, so there is no need to perform an impedance matching design in the PCB layout. Still, you need to pay attention to the 75 ohm grounding resistance in the circuit that should be placed close to the terminal pin.

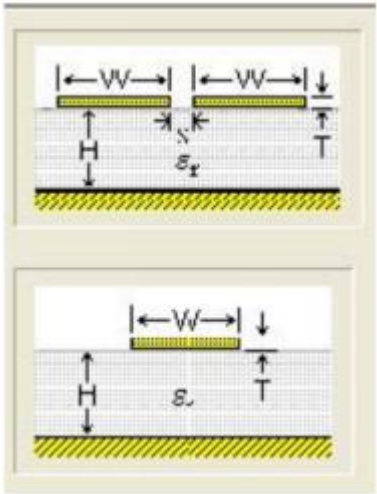
PP types

Type	Dielectric thickness	Adjustable range	Dielectric constant
1080	2.8mil	2.0-3.0mil	4.3
2116	4.2mil	4.0-5.0mil	4.3
1506	6.0mil	5.5-6.5mil	4.3
7628	7.2mil	7-8.5mil	4.3

Solder mask thickness: 0.6±0.2mil

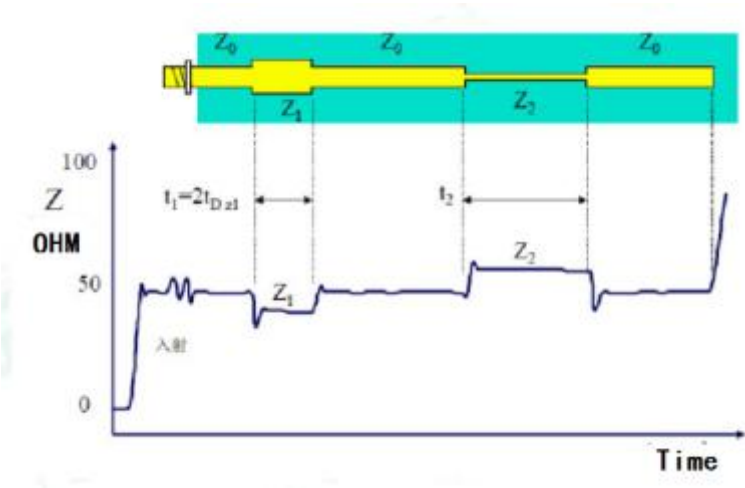


Cer=3.5+/-0.3



Impedance Test:

Impedance test receives the reflected wave simultaneously after the oscilloscope sends out a pulse wave, then compares and analyzes the two pulse waves, and gets the impedance value from the reflected energy.



Impedance test schematic

Using TDR (Time Domain Reflectometry) measurement, the general measurement method uses “time domain reflectometry.”

The calculation formulas of some impedance types are introduced above, but the current impedance is calculated by the software, which is faster and more accurate. Our factory currently uses Polar’s test instrument CITS500s.

Test range: 0-300ohm

Test accuracy: 1%—50ohm

1.25% — 75ohm

1.5% — 28\100ohm

Reflected Pulse Rise time: ≤200ps

System bandwidth: 1.75GHz

The dielectric constant Er will change with the working frequency. Within a specific range, the higher the frequency, the smaller the Er. Annex I, Annex II, and Annex III are core board Er provided by three sheet suppliers, which can be viewed at 1MHz. The dielectric constant is greater than that at 1GHz.

- Characteristic impedance

It is composed of resistance and reactance (combined by inductive reactance and capacitive reactance). The characteristic impedance in the PCB depends on the width and thickness of the wire, the distance between the wire and the ground plane, and the dielectric constant (ϵ_r) of the medium between the wires.

- Impedancematch

The signal transmission in the electronic circuit starts from the output of the power supply. It expects to be transmitted to the receiving end without energy loss and any signal reflection in the middle.

Therefore, the impedance (ZL) in the PCB is required to be equal to the impedance (ZO) of the power supply terminal, which is called impedance matching. If the impedance cannot be matched, the received signal is distorted.

- Micro-strip

A transmission line structure in which the wires are parallel to the ground plane in the PCB and separated by the medium.

- Strip-line

A strip-line is a high-frequency transmission wire between two parallel ground planes (or power planes) between dielectrics.

[Request Impedance Control PCB Quote](#)

PCB Impedance coupon design

In the circuit, the ratio of voltage to current in the transmission wave between two points, including the [assembled components](#), is the resistance of any point of the transmission line to the transmission wave.

- Square PAD “■”——The outer layer of PTH corresponds to PAD, and the inner and outer layers of this PTH are only connected to the relevant shielding layer.
- RoundPAD “●”——The outer layer of PTH corresponds to PAD, and the inner and outer layers of this PTH are only connected to the signal lines of the relevant layer.
- A font is added next to each outer round pad and square pad to indicate the layer sequence of the signal line layer or shielding layers. For example, 1, 2, 3, 4, 10, and 11 represent the signal layer or shielding layer in the first, second, third, tenth, and eleventh layer.
- Except for the gong pipe position hole, the other holes are made of PTH, and the upper pad or round pad is added accordingly.
- When there is more than one impedance baron Wpnl, add different numbers 1, 2, 3, 4, etc., to the circuit layer/silkscreen layer for identification during testing and analysis.
- The PTH hole diameter of each impedance baris uniformly $\phi 1.00\text{mm}$, and two gong pipe holes $\phi 1.60\text{mm}$ should be added.
- When there is a need for production layout or additional control of impedance lines is required, a single-hole connection signal line design can be used. See P4 and P5 diagrams to save design positions.
- Suppose the impedance bar is designed in a range with a small length (sometimes, the customer will specify the location). In that case, the signal line can be in a turning form to ensure the total signal line. It is important to avoid right-angles in wiring and maintain a large turning radius. The length is sufficient (refer to the



tenth requirement at the same time). Still, the signal line distance should be kept above 2.54mm. See figure P8.

- If there are no special requirements, the gap between the signal line and the DummyPattern is five times the minimum distance (mm) to the shielding layer.

For non-coplanar type impedance bars, if the customer does not specify requirements when designing the impedance bar, the Dummy Pattern next to the signal line (the distance between the long copper bar or copper pad and the signal line, t , must be $\geq 5H$ (H is the signal layer to the shield) The insulation thickness or minimum thickness of the layer). The following examples illustrate.

Note: For non-coplanar type impedance, when the distance between the Dummy Pattern and the copper skin is less than $5t$, the impedance will become smaller as the distance becomes smaller, even if other conditions have not changed.

A. For Microstrip type, t = insulation thickness from the signal layer to the shielding layer.

B. The Stripline type, t = the smaller of the insulation thickness from the signal layer to the two shielding layers $H1$ and $(H-H1-T)$. t = the insulation thickness min.

*Differential impedance is handled in the same way

Note: Not only the distance between the signal line and the Dummy Pattern in the same layer must be made according to the above requirements, but also in the situation in the following figure. When L3 has impedance control requirements, the signal line of the L3 layer is in the other layer (the figure is L4 as an example). The spacing t of Dummy Pattern should also be made according to the above requirements.

As for the impedance of the coplanar type, the designer (customer) specifies the spacing. The spacing between the signal line and the copper coupon next to it – is a component of the coplanar type impedance, which cannot be agreed upon by the customer and cannot be changed. To ensure proper spacing, the next copper wires or large copper bars should also be vibrated thick.

- Single-line or double-line impedance, each with 2 or 4 test holes, must ensure that the relative distance between their hole centers remains unchanged (see P); otherwise, the test cannot be performed. It is 2.50mm), the hole size is 0.8mm to 30 mm (refers to the customer to design. Please refer to the fifth requirement for factory design). If the customer has designed a test pattern in the PCB, the relevant PPE personnel will check whether the design is suitable for the test instrument requirements of the factory. If it is not suitable, consult the customer to change the design to facilitate the [factory test](#). If there are disagreements, one can also design a coupon to be able to test.
- In general, the length of the signal cable should be 6" or more.
- If there are no special instructions, for all large copper skins, the inner layer is 0.50mm from the outer shape of the Coupon, and the outer layer is 0.30mm from the outer shape of the Coupon.



- For the pads added by the inner and outer layer to the [PTH](#), ensure that AR=0.25mm. For the inner layer, cut out copper for non-connected holes, ensure that Clearance=0.30mm (all layers are also made of copper for gong pipe positions). Solder the solder mask Pan and gong pipe position hole plus window, one side larger 0.10mm.
- Vibration according to MI and film production instructions.
- Unless the customer requires, do not design the impedance bar in the Unit/Set.
- Place the impedance bars close as possible to the control line in the PCB. This is to make the control line on the impedance bar after plating and etching have better consistency with the copper thickness and line width in the PCB. Arrange this position in the middle area of the layout.
- See P — P for standard graphics. According to different line width, impedance, and pressure plate requirements, select the corresponding graphic combination and make the corresponding connections and adjustments, and the design will be completed.

The current analysis of the impedance bar is to slice at three positions at both ends and the middle and detect the following: copper thickness, the upper and lower top widths of the circuit, and the average value of the insulating layer thickness (some need to add the thickness of the solder mask). Then add the measured impedance value and compare it with the designed theoretical value to determine whether the impedance is okay.

Impedance control difficulties

1. Medium thickness

Uneven pattern distribution results in an uneven thickness of the medium, fluctuations in impedance, and discontinuity.

2. Line width

The line width/line spacing is getting smaller and smaller, and the line width accuracy is getting more challenging to reach $\pm 10\%$.

Outer layer electroplating unevenness – The copper thickness is affected by electroplating parameters, uneven pattern distribution, hanging board method, and other factors. The thickness of different patterns varies greatly, resulting in inconsistent line width.

The future development trend of impedance control

1. Higher impedance accuracy and smaller impedance fluctuation.
2. The control accuracy requirement is increased from the current $\pm 10\%$ to $\pm 7\%$ or even $\pm 5\%$.
3. Higher line quality requirements.
4. More stringent requirements for line gaps, residual copper, pits, and other defects.
5. Lower line roughness.
6. Small burrs, larger etching factor.
7. [High-speed](#) impedance transfer layer via design.

Related Posts:

- 1. [How to do PCB Impedance calculation model encountered in PCB design](#)
- 2. [PCB Testing Laboratory](#)
- 3. [Resistor Color Code Calculator and Chart \(4-band, 5-band or 6-band\)](#)
- 4. [The Impedance of FPC Circuit Board](#)

Recommended Posts

14 Layer PCB

Basic Skills about PCB etch

What is Nan Ya PCB?

Two types Soldering Dip and SMD



We'd love to hear from you

Name



SEND MESSAGE

Connect with us

