

IMPEDANCE IN PRINTED CIRCUIT BOARDS

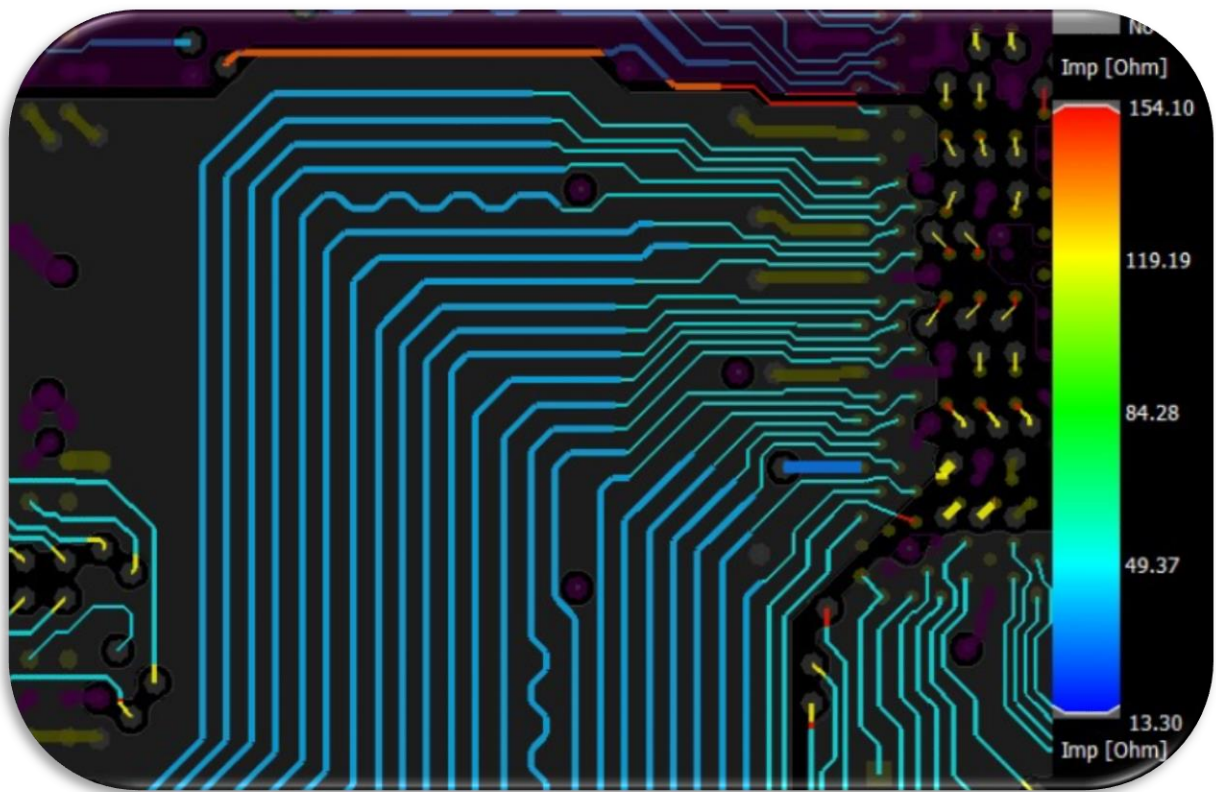


Image Source – Cadence

What is Impedance?

The term impedance describes the behaviour of electronic components in resisting AC current. A circuit is made up of various components that behave differently in the presence of an AC source.

The resistor behaves consistently regardless of the frequency of the AC source, as it is a pure DC component. Its impedance is a measurement of its resistance:

$$Z = R$$

Meanwhile, the capacitor's impedance equals its capacitive reactance, which is given by:

$$X_c = 1/2 \pi fC$$

Capacitive reactance is an inverse function of the angular frequency. As the frequency increases, the capacitive reactance decreases.

As for inductors, they behave in a similar way with capacitors, except that inductive reactance is directly proportional to the angular frequency, with the formula:

$$X_L = 2\pi fL$$

While these are basic components often associated with impedance analysis, they are not the only ones present in a circuit. In a design, you are likely to have ICs, diodes, transistors, and other passive components, and they may introduce impedance to the circuit.

At a deeper level, your traces have some impedance, as they cannot be considered as long inductors. The board itself will affect the impedance of a circuit and your traces. The insulating PCB substrate creates parasitic capacitance, while the arrangement of traces and planes in interior layers creates parasitic capacitance and inductance. These parasitic effects contribute to capacitive crosstalk, and they determine the impedance of transmission lines and your power delivery network.

What is controlled impedance?

Impedance is simply the degree of opposition to the flow of energy in an electrical circuit or a transmission line.

It is denoted as Z and measured in Ohms.

It is the result of summing the resistance (R) and the reactance (X) of an electrical circuit.

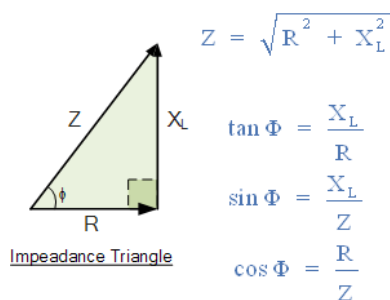
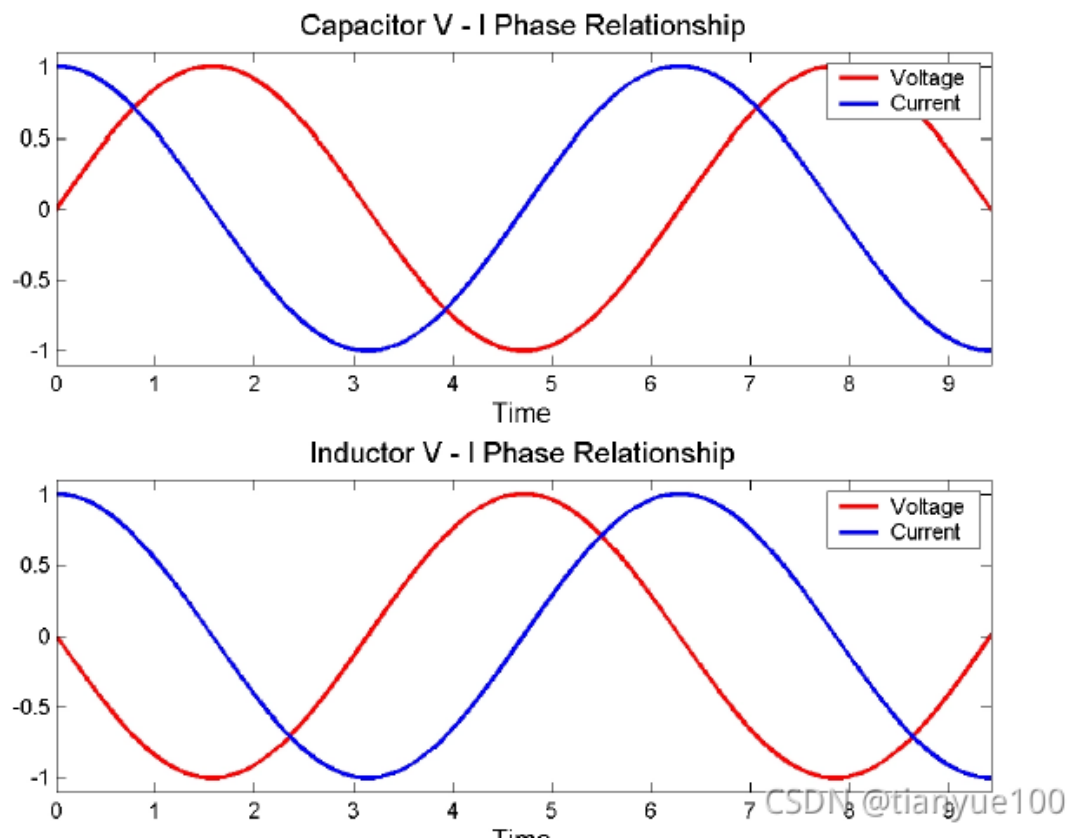


Image source – Electronics Tutorials

Inductance (L) which is the induction of voltages in conductors due to the magnetic fields of currents.

Capacitance (C) which is electrostatic charges storing due to the voltages among conductors.

- Inductance (L) which is the induction of voltages in conductors due to the magnetic fields of currents
- Capacitance (C) which is electrostatic charges' storing due to the voltages among conductors.



Difference between Impedance and Resistance

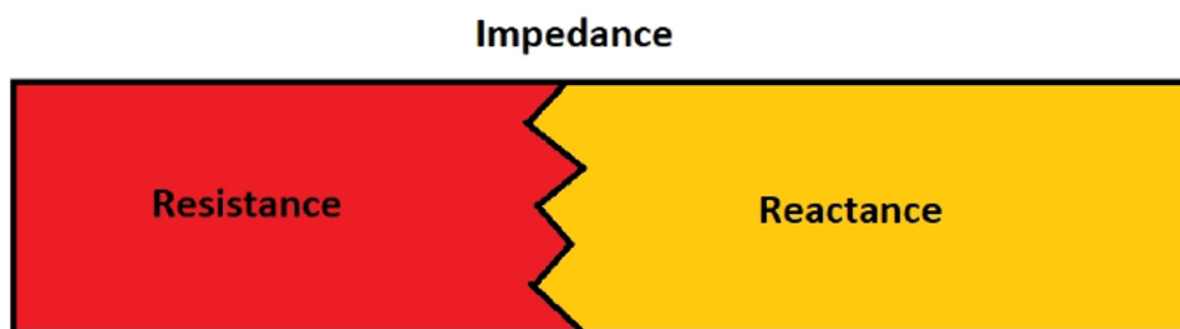


Image Source – Stack Exchange

Please note that Ohm's law is the first equation.

$$v_R = iR$$

$$v_L = iZ_L$$

$$v_C = iZ_C$$

$$\text{Inductive reactance: } X_L = 2\pi fL = \omega L$$

$$\text{Capacitive reactance: } X_C = \frac{1}{2\pi fC} = \frac{1}{\omega C}$$

When $X_L > X_C$ the circuit is Inductive

When $X_C > X_L$ the circuit is Capacitive

$$\text{Total circuit reactance} = X_T = X_L - X_C \text{ or } X_C - X_L$$

$$\text{Total circuit impedance} = Z = \sqrt{R^2 + X_T^2} = R + jX$$

Image source – Electronics Tutorials

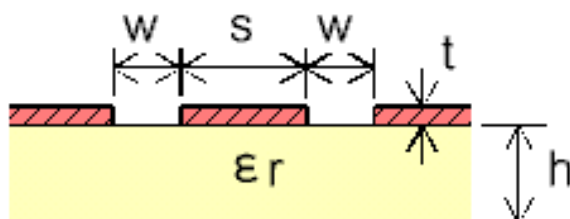
Characteristics of Impedance in Printed Circuit Boards (PCB)

A PCB trace has several characteristics to consider regarding impedance;

These include.

- Height - H
- Width - W
- Spacing - S (For Differential Pair Signals)
- Distance between the track and other copper features (including copper layers beneath or on top of the signal layer containing controlled impedance)
- The dielectric constant - ϵ_r
- PCB fabrication tolerances/limits etc.

All of these are characteristics to consider when calculating impedance and manufacturing Controlled Impedance PCBs.



Why You Need Controlled Impedance PCB

- Need for More Signal Power transmission.
- Improved Performance.
- Control energy flow.
- Need to manage electromagnetic interference – EMI.

Why Impedance of single ended signal is 50 Ohms?

In the process of designing the PCB, before the routing, we generally laminate the items that we want to design, and calculate the impedance based on the thickness, substrate, and number of layers. After the calculation, the following contents are generally obtained.

Finished Thickness (mm) : 1.2 ± 0.12				
Account Thickness (mm) : 1.14				
LAYER STACKING				
TOP			0.5oz +Plating	positive
	PP(3313)	3.63		
GND02			1oz	negative
	Core	5.12		
ART03			1oz	positive
	PP(7628*3)	20.92		
ART04			1oz	positive
	Core	5.12		
PWR05			1oz	negative
	PP(3313)	3.63		
BOTTOM			0.5oz +Plating	positive
SINGLE IMPEDANCE :				
	layer	width(mil)	impedance(ohm)	ref layer
	L1/6	5.50	50.00	L2/5
	L3/L4	6.50	50.00	L2/5
DIFFERENTIAL IMPEDANCE:				
	layer	width/space(mil)	impedance(ohm)	ref layer
	L1/6	4.1/8.5	100.00	r=
	L3/L4	4.5/8.5	100.00	BE China Comm

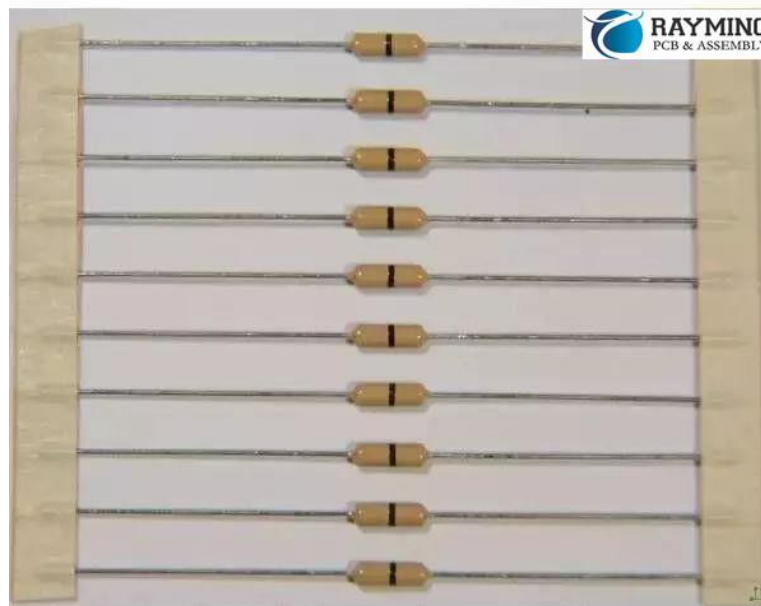
Image source – RAYMING

As can be seen from the above figure, the single-ended network above is generally controlled by 50 ohms. Many people will ask why it is required to control 50 ohms instead of 25 ohms or 80 ohms.

First of all, the default choice is 50 ohms, and everyone in the industry accepts this value. In general, it is definitely a certain standard that has been developed by a recognized institution. Everyone is designed according to standards.

A large part of electronic technology comes from the military. The first technology is used in military applications, and it is slowly converted from military to civilian. In the early days of microwave applications, during the Second World War, the choice of impedance was completely dependent on the need for use, without a standard value. As technology

advances, impedance criteria need to be given in order to strike a balance between economy and convenience.



EEChina.com

In the United States, the most used conduits are connected by existing gauges and water pipes. 51.5 ohms are very common, but the adapters and converters seen and used are 50-51.5 ohms; these are solved for the joint army and navy. The problem was that an organization called JAN was established (later the DESC organization), which was specially developed by MIL. After comprehensive consideration, it finally chose 50 ohms, and the related conduits were manufactured and converted into various cables. Standard.

At this time, the European standard was 60 ohms. Soon after, under the influence of the industry-dominant companies like Hewlett-Packard, the Europeans were forced to change, so 50 ohms eventually became a standard in the industry. It became a convention, and the PCB connected to various cables, in order to match the impedance, was finally required according to the 50-ohm impedance standard.

Secondly, the formulation of general standards will be based on the comprehensive consideration of PCB production process and design performance and feasibility.

From the perspective of PCB production and processing technology, it is relatively easy to produce a 50 ohm impedance PCB considering the equipment of most existing PCB manufacturers. From the impedance calculation process, it is known that too low impedance requires a wide line width and a thin medium or a large dielectric constant, which is difficult to satisfy spatially for the current high-density board; too high impedance requires a thin line. Wide and thick media or a small dielectric constant is not conducive to the suppression of EMI and crosstalk, and the processing reliability will be poor for multi-layer boards and from the point of view of mass production. Control 50-ohm impedance In the environment

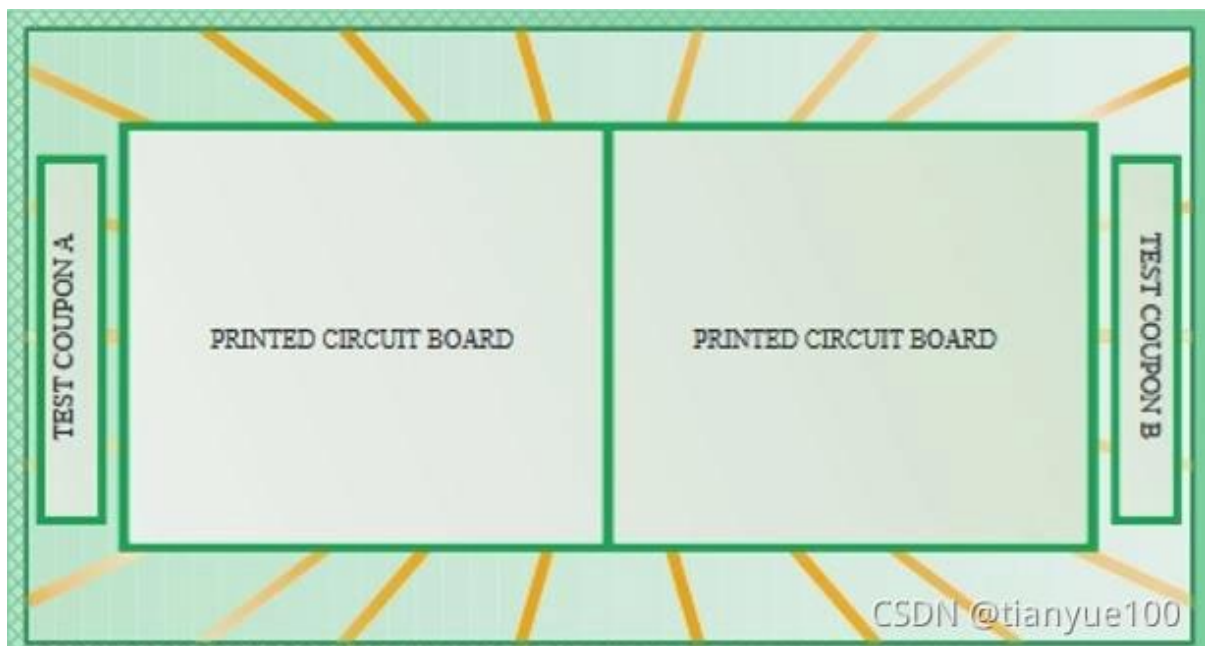
of common board (FR4, etc.) and common core board, produce common thickness products (such as 1mm, 1.2mm, etc.), and design common line width (4~10mil). The factory is very convenient to process, and the equipment requirements for its processing are not very high.

From the perspective of PCB design, 50 ohms is also chosen after comprehensive consideration. In terms of the performance of the PCB trace, the general impedance is relatively low. For a transmission line with a given line width, the closer the distance is to the plane, the corresponding EMI will be reduced, and the crosstalk will be reduced. However, from the perspective of the full path of the signal, one of the most critical factors to consider is the driving capability of the chip. In the early days, most chips could not drive transmission lines with impedance less than 50 ohms, and higher impedance transmission lines were inconvenient to implement. Therefore, the compromise uses 50-ohm impedance.

Therefore, 50 ohms is generally selected as the default value of the conventional single-ended signal control impedance.

Testing and measuring Controlled Impedance PCB

Most Controlled Impedance PCBs undergo testing before they are put to use in a project. Due to this, testing is not usually done on the PCB itself, but one or two test coupons integrated into the PCB panel.



Typical Production Panel

Note that coupons can often be standard in shape, size, and probe pinout, among other things. Standardization, in this case, is to allow the fabricator to build test fixtures that will facilitate-and accentuate-testing. During the design stage, you can have coupons as part of the main board. To test, the coupons are inspected to ensure proper layer alignment, electrical conductivity, and cross-sectioned to examine internal structures. If you need an accurate test for impedance, you can ask your manufacturer to design a test coupon separately. To test, the coupons are inspected to ensure proper layer alignment, electrical conductivity, and cross-sectioned to examine internal structures. If you need an accurate test for impedance, you can ask your manufacturer to design a test coupon separately.

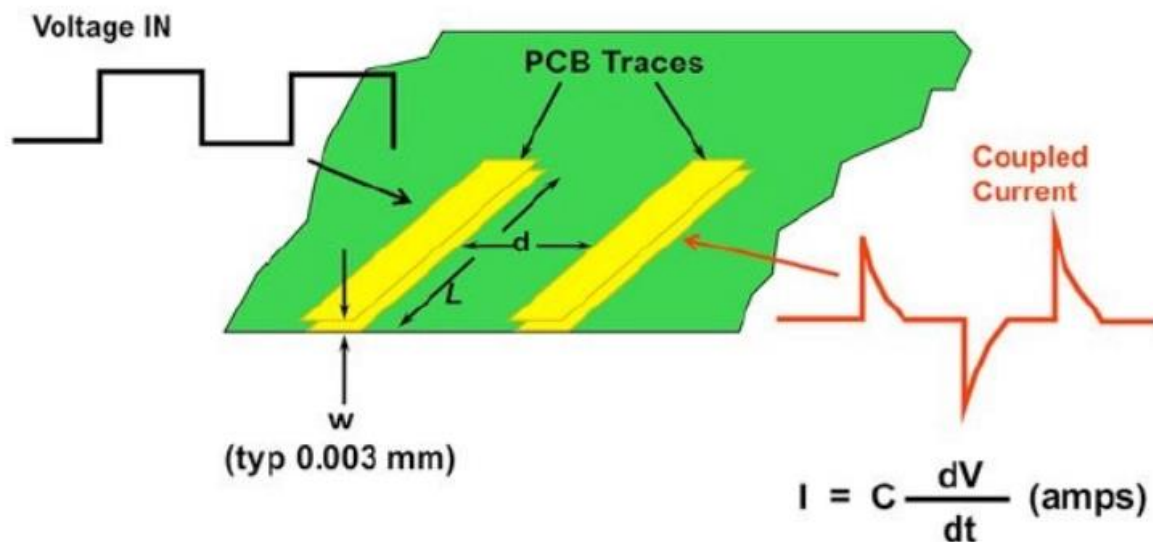


Image source – Signal coupling from EDN

Impedance is then tested by using a TDR (time-domain reflectometer).

To measure impedance, the TDR applies a fast voltage step to the test coupon via controlled impedance cable and probe. Any reflections that occur on the waveform will show on the TDR, including the value of discontinuity. Discontinuity, in this case, refers to a change in impedance value. So, if there is a discontinuity, the TDR will display its location and magnitude.

Please note;

The overall performance and EMC behavior of electronic equipment are not just determined by the circuitry and geometry of the layout. But also, by the power distribution network (PDN).

In this case, you need to pay careful attention to.

- The choice of decoupling capacitors and quantity required and routing loops.
- The plane capacitance required by different voltages to accommodate noise limits.
- Reference plane continuing and return current paths.
- Inductances caused by poor component packaging.

Apart from using controlled impedance TDR techniques, you can also measure impedance using a network analyzer or a laboratory TDR.

- The TDR Oscilloscope waveform will display the reflections created by the impedance discontinuities.
- Measurements can be made to determine how much the impedance deviates from the nominal value.

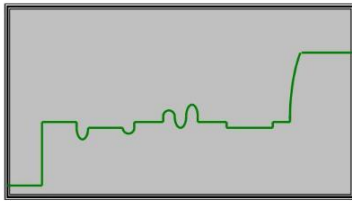


Image source – Textronix

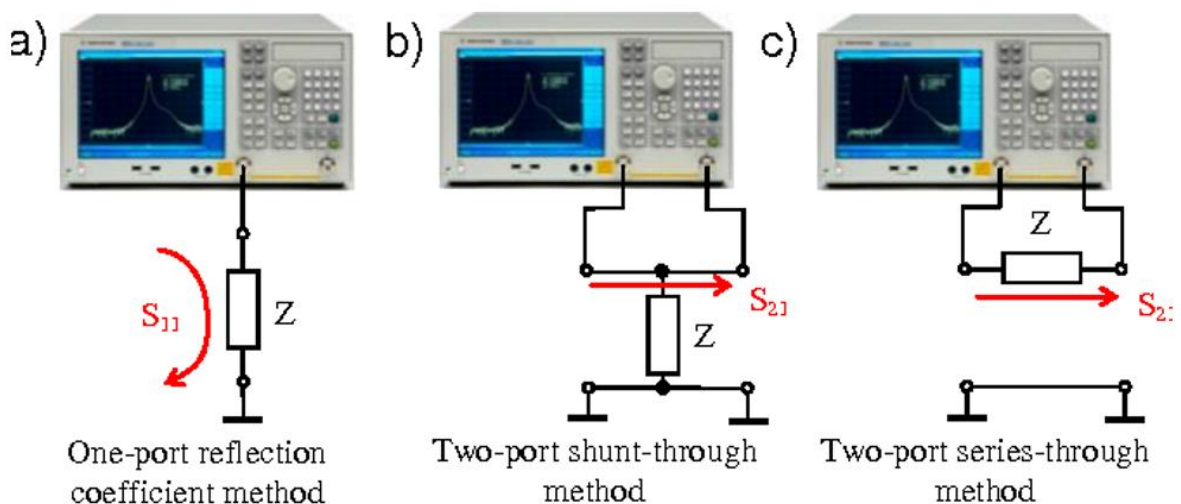


Image source – Impedance measurement NA – Semantic scholar

Calculating Controlled Impedance in Printed Circuit Boards

Understand that the impedance of a PCB is primarily influenced by;

- The distance of the signal layer
- Conductor geometries
- Trace width
- Copper thickness
- Permittivity ϵ_r

Now, you can calculate controlled impedance using simple equations to obtain nominal values of trace dimensions for a particular impedance.

Calculation of PCB Track Impedance (polarinstruments.com)

<https://www.polarinstruments.com/support/cits/IPC1999.pdf>

PCB signal integrity, stackup and controlled impedance (polarinstruments.com)

I recommend that you use a PCB impedance calculator to calculate PCB controlled impedances.



PCB stackup, signal integrity and controlled impedance

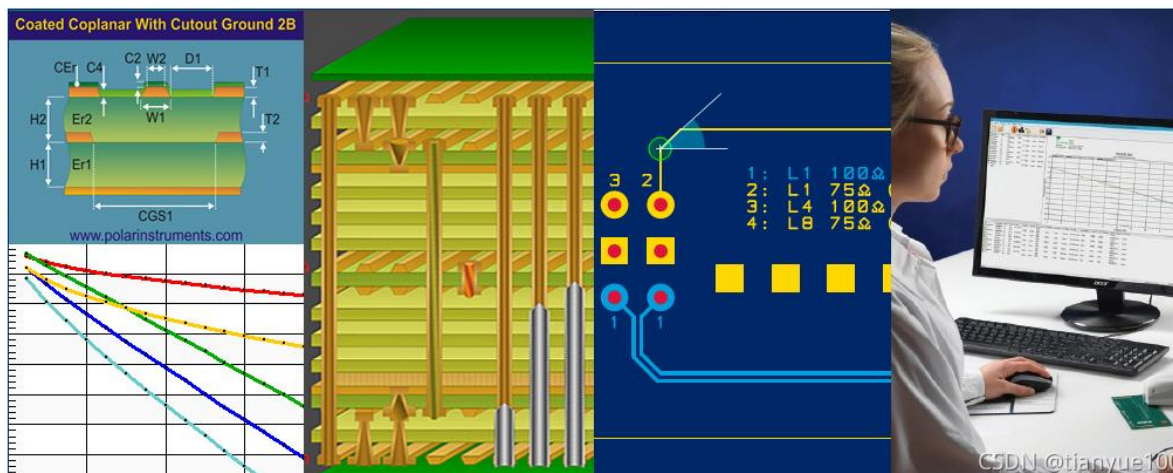


Image source – Polar Instruments

How to Specify PCB Impedance

Usually, PCB manufacturers will offer a standard stack up.

2-Layer Aluminium-Carrier

Top L1	Solder mask		15µm	Lacquer
	Outer layer		35µm	Copper
			100µm	FR4
IL2	Innerlayer		35µm	Copper
	Prepreg		100µm	1x2116 FR4
	Aluminium		1500µm	Alloy 5052 - 1100
Total thickness			1,79mm	
Tolerance			±10%	
Maximum thickness			1,97mm	
Minimum thickness			1,61mm	

You will use this to calculate, either by hand or software, what the trace dimension should be based on this stack up. If you find the results plausible, use that. If not, you'll need to specify a stack up that will work for you.

How do you do this?

Well, first of all, start with a trace thickness that is practical for routing, spacing, manufacturability, etc. Then, calculate the dielectric thickness, given material with specific dielectric constant, for the impedance you need.

Levels of Impedance Control in PCBs

Now that you know the basics of impedance control, it is essential also to know the different levels of impedance. This knowledge is handy when deciding what kind of impedance control service, you need for your PCBs. With that in mind, there are three levels of impedance control. These are:

1. Impedance Control

Impedance control popularly applies to high-end designs with tight tolerances or unusual configurations.

It is best used if your design has tight impedance tolerances, as I've said, that could be tough to hit the first time around. As you will learn later in this guide, there are different types of controlled impedance.

There is characteristic impedance, which is the most common, and then there are.

Characteristic Impedance

Characteristic impedance (Z_0) is the most important parameter for any transmission line.

From: The Circuit Designer's Companion (Fourth Edition), 2017

- Wave impedance
- Image impedance
- Input impedance

In the case of impedance control, your manufacturer will build the board. Then he will test it via TDR to see if it meets initial impedance specifications.

2. Impedance Watching

Impedance watching refers to a situation where the impedance control trace is indicated on the design. Here, the designer will just outline the impedance control trace. The PCB supplier will then adjust the trace width and dielectric height as needed. Upon approval of the complete specifications, the manufacturer can begin to build the board. If your manufacturer allows, you can request for a TDR test to confirm the impedance for a small fee.

3. No impedance Control.

If your design does not have tight tolerances, a no impedance control service would be ideal. In this case, you won't need any extra design elements to ensure correct impedance. Instead, you can achieve correct impedance by conforming to standard specifications without impedance control. Your manufacturer can provide accurate impedance without special measures, which make this the more cost-effective option.

Control Impedance Manufacturing Process

Unfortunately, we can't discuss all of these steps independently. So, for the scope of this guide, I'll discuss the manufacturing process in at most three categorical stages.

Stage 1: Controlled Impedance PCB Design and Layout



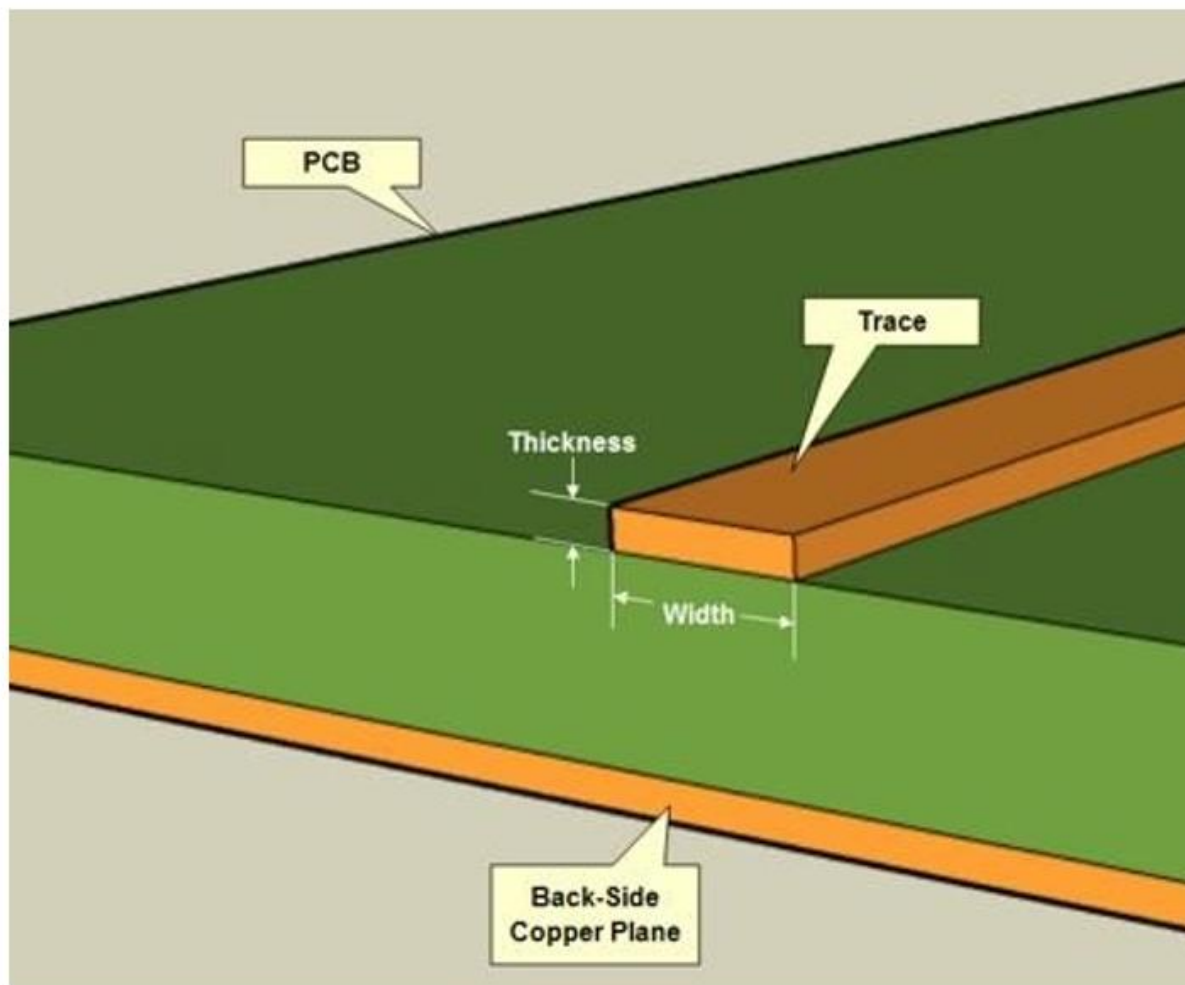
Before fabricating a PCB, a manufacturer must first define its design vaguely. And being an intricate design, the first step would be first to define how to control impedance.

How to Control Impedance

As you can tell, fabricating a Controlled Impedance PCB is a task. This is because it requires a high level of care to achieve consistently accurate results. So, you should know that the design is only the first challenge. Fabrication, in its entirety, must be completed with a well-understood process. Etching, for instance, must be accomplished without under or over-etching. The substrate is also the dielectric in this case. Thus, it must be held to a reasonable tolerance to assure the expected impedance. Now, when controlling impedance, you must make sure the impedance is constant at each point along the trace. For this, you'll need to control three key features of the circuit's geometry. These are the trace width, the spacing between the signal return path and the signal trace. It also includes the dielectric coefficient of the material surrounding the trace as well as the trace thickness. You can change these features and still retain controlled impedance.

This is as long as you change other features as necessary, so the relationship between these aspects does not change, and impedance remains constant.

Controlled Impedance Design Considerations



As I've said, controlling impedance means maintaining a constant trace impedance at every point of a PCB trace. Meaning, wherever the trace travels, even if it changes layers, the impedance should stay the same throughout. That is, from start to end.

In Controlled Impedance PCB manufacturing, there is usually little control over the impedance in a device driver or the load. Nonetheless, you can control the impedance on the PCB. As such, you have to match the circuitry on the PCB to the impedance of the source and load. This way, you can ensure a consistent appearance throughout the entire path of a signal. In this case, essential design elements that you have to consider alongside proper design techniques are:

Choice of Materials

Effect of Fiber Weave Structure in Printed Circuit Boards on Signal Transmission Characteristics

https://www.researchgate.net/publication/330537816_Effect_of_Fiber_Weave_Structure_in_Printed_Circuit_Boards_on_Signal_Transmission_Characteristics

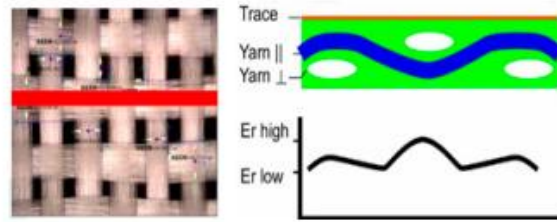


Figure 2. The Dk variation due to fiber weave.

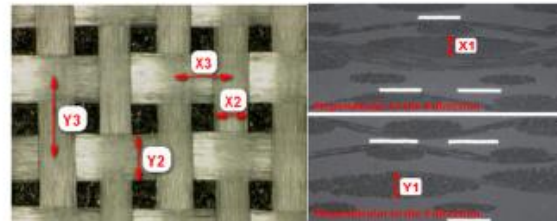


Figure 3. Photos to illustrate fiber weave bundle parameters.

Table 1. Measurement results of fiber weave bundle parameters.

Style	Measurement Results (mil)					
	X1	X2	X3	Y1	Y2	Y3
1035	0.82	8.8	14.2	0.78	12.4	13.7
1080	1.6	8.2	17.0	1.1	12.1	22.4
1078	1.4	14.2	16.2	1.0	17.6	17.8
3313	1.9	13.1	16.2	1.5	11.0	16.3
2116	2.2	14.1	17.2	2.0	15.5	17.3

$$\Delta\text{Phase} = \text{Skew} * \text{Frequency} * 360.$$

(4)

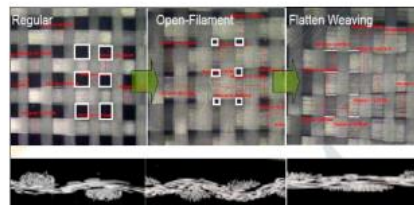


Figure 11. The structure of regular weaves and flattened fiber weaves.

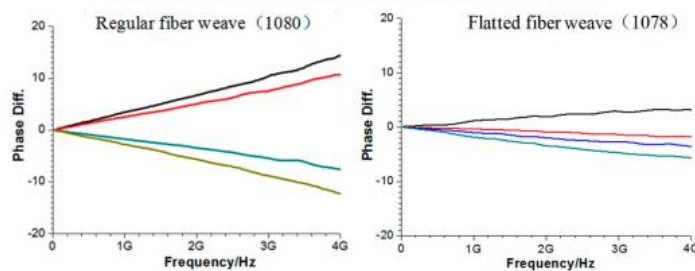


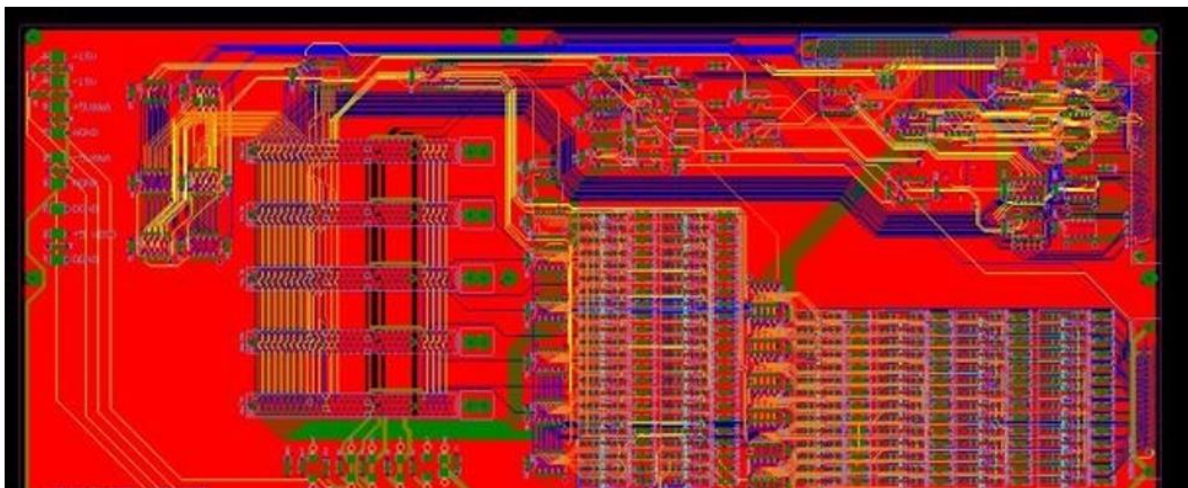
Figure 12. ΔPhase of regular and flattened fiber weaves.

Before, FR4 was commonly used in PCB fabrication. But with the advent of high-speed designs, correct laminates must be used. Here, you'll need to specify the use of a material with a lower dielectric constant. This will help ensure best signal performance as well as minimize any cases of signal distortion or phase jitters. Another thing that you need to consider in this is the loss tangent, which is also known as the dissipation factor. It refers to

the measure of signal loss as the signal propagations down the transmission line on the PCB. For high-speed designs, you may want to choose the lowest loss material. Note that different laminate materials have different loss tangents. As such, you need to choose the material that is most suitable for your application and inform your manufacturer accordingly. Additionally, you need to take into account the weave pattern when choosing the PCB laminate material.

You see, typical PCB core and pre-peg substrates are built from various woven fiberglass fabrics. These are bound together with epoxy resin. The fiberglass and epoxy each have distinctive dielectric constant values. This prompts an inhomogeneous mechanism for signal propagation. A loose weave pattern will, in general produce less uniform dielectric constants in a PCB overlay. This can result in trace impedance variations, and propagation skews. And the higher the speed, the more evident this problem will be. On the other hand, a tighter weave pattern means a more uniform dielectric constant. Therefore, it is crucial to choose a tighter weave pattern for the signal to be able to move over more glass. This will result in a highly consistent dielectric constant throughout the board.

Power Planes



PCB Design and Power Plane

In this case, the circuit won't have any power islands to support the voltage. So, if you're designing a high-speed circuit, it is important that you use lots of decoupling capacitors. As you do this, remember to give special care to the RF and power supply switching sections' grounding planes. For these, you'll need to isolate their islands from the system ground plane.

You must also include track connections switching island to system ground. That is, the tracks ought to be huge enough to have close to zero DC resistance, yet not more. The reason for doing this is to avoid switching RF sections. This can create waves on the ground plane that can create ground bounce on the system ground. You can read this article on PCB power-supply design if you need more explanation on this subject.

Other things that you should keep in mind when designing a Controlled Impedance PCB include.

- Maintaining shorter trace lines.
- Avoid routing stubs and discontinuities.
- Maintain same lengths on signal pairs for differential pair routing.
- Use back drilling to remove unwanted copper.
- Use immersion silver as a surface finish instead of ENIG.
- Use smaller anti-pads on plane layers.
- Always specify the solder mask thickness.

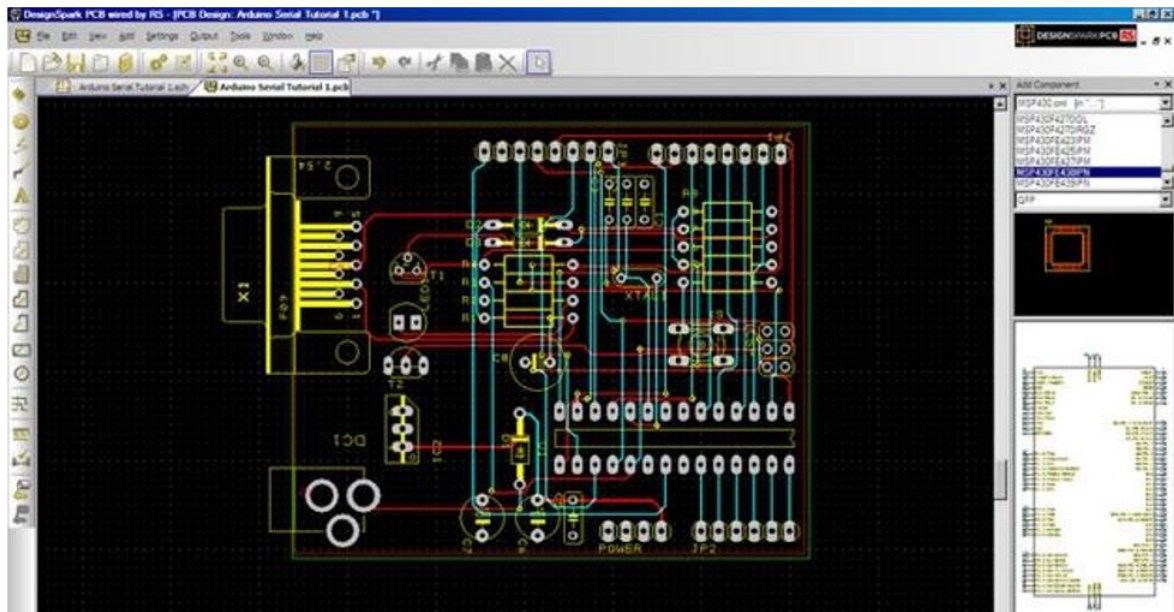
These are the main factors that you need to consider when designing a Controlled Impedance PCB. Note that you should integrate these during the design and layout stage. When done, check to verify if the design is okay and as per your requirements or not and rectify where necessary. Remember that it is always easier and cheaper to rectify a PCB design before the board is manufactured.

Common Controlled Impedance PCB Design Mistakes

- During design analysis, the following mistakes are often encountered.
- Traces are crossing split lanes. Signals should always be routed on solid ground reference planes and not across a split plane or void in the reference plane.
- Traces without a reference ground plane. Impedance is often high if there are no adjacent layers. As such, it is advisable to route high-speed signals on the top or bottom layer of the board.
- Mismatches in length. This can lead to signal distortions and an increase in bit error rate.
- It is thus advisable to length-match differential pairs ± 5 mils of each other if possible.
- Use of too much pre-pegs. It is inadvisable to use more than three different types of pre-pegs in a stack up.
- Wide impedance trace space. The spacing between two traces of a differential pair should never exceed twice the width of the traces.

If you're intent on designing a functional Controlled Impedance PCB, avoid these mistakes at all costs.

How to design controlled impedance PCB



Stage 2: Controlled Impedance PCB Prototyping

After the design and layout of a Controlled Impedance PCB, a prototype is made. Again, this is before the actual fabrication of the commercial board. The prototype in this stage plays a key role in the creations of a PCB design. It facilitates the manufacturer to foresee if anything needs to be resolved in the PCB design. In case of any failure in the prototype, a new prototype is created and is kept under custody until it performs well.

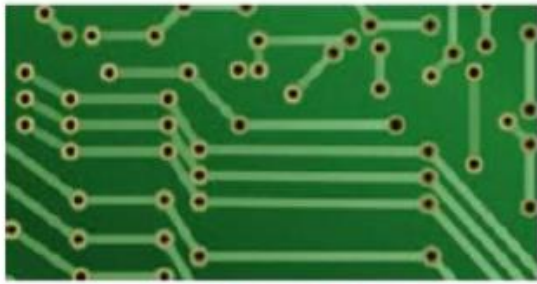
Stage 3: Controlled Impedance PCB Assembly

Once approved, the prototype proceeds to the next stage, which is actual fabrication and assembly. The board is first fabricated to specifications. It is then connected with electronic parts and components as specified in the layout stage. We call this PCB assembly. PCBA

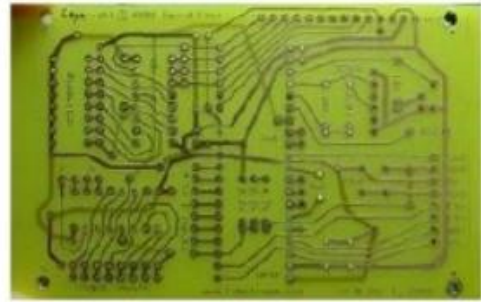


PCB Assembly

Types of Controlled Impedance PCB



Single Sided PCB



Double Sided PCB

1. Single sided Controlled Impedance PCB

In PCB design and fabrication, single-sided types are the least complex. These are built with all electrical parts attached to one side of the base material, and with other side coated with copper traces. Copper, in this case, is the preferred type of metal because it is a very effective electrical conductor. A special solder mask often protects the copper layer. Also, a silkscreen coating is a further feature to help mark the different components on the board. Single-ended controlled impedance boards are preferred for the most basic electronic items. They're also often the first type of boards used by at-the home hobbyists. Note that although single-ended boards are the most cost-effective to the manufacturer, they aren't the most used. This is because of their design and uses limitations.

2. Double-sided Controlled Impedance PCB

Double-sided controlled impedance boards are a standard choice for a wide range of applications. They're built with components, and parts mounted to both sides of the base material. This type of board is designed with plenty of holes to make it possible for the circuits on each side to connect. The wires are soldered in place to give a strong and reliable hold. A further option to connect the two sides is through-hole technology. This technology can create devices that run at faster speeds and less weight. They use small leads that are permanently soldered to the board instead of using separate wires.

3. Multi-layered Controlled Impedance PCB

Multi-layer PCB design

The multi-layer-controlled impedance boards are made up of several base materials with each separated by insulation. Standard sizes for this board type include regions with 4, 6, 8, and 10 layers. However, it is possible to manufacture huge boards with up to 42 layers or even more. Such large sizes are mostly preferred for more complex applications. The different boards in multilayer PCB designs are connected using wires passing through the individual holes. They help minimize issues with space and weight. Applications for this type of board range from handheld devices, space probe equipment to medical machinery, and servers. Controlled impedance circuit board manufacturing can be carried out for a limited number of circuits. Or, for large volume production. It is therefore essential that you choose

a Controlled Impedance PCB manufacturer with a proven track record. Read on to find out how.

How to Choose Controlled Impedance PCB Manufacturer

Having learned all that, you have in this guide, it should not be difficult to find a quality Controlled Impedance PCB for your project. You know what to look for, what to ask for, and what to reject, etc. The problem is, who you can trust to build you a great board for your application? You see, there are several manufacturers worldwide who have expertise in designing and producing Controlled Impedance PCBs.



So, what are the aspects to be taken care of when selecting a controlled impedance PCB manufacturer?

Use of Modern Technology

The approach of new innovation has refreshed everything, and nobody would need obsolete and old software design. In that capacity, it is smarter to choose a manufacturer that use the latest designs to make the circuit work.

Experience and Reputation

It is exceedingly vital to choose a manufacturer with great experience in producing Controlled Impedance PCBs. It is essential to check if the manufacturer has a skilled team of designers and engineers who are familiar with the intricate details of controlled impedances sheets. It is better to choose a manufacturer that boasts of at least 10 years' experience in this area.

Receptiveness to Customization

An unbending nature in design is the last things any sane buyer would acknowledge. Therefore, it is crucial to work with a manufacturer that is ready to tweak and work as per requirements.

Timely Delivery

Time is of the essence in any business transaction. Thus, the manufacturer you contract should be able to meet deadlines and deliver your order on time.

Servicing

Repairing circuit boards is relatively easy. Nonetheless, it is nearly impossible for everyone to know how to do it. In such cases, the manufacturers should be ready to provide apt servicing on the off chance that you need it.

Quality Control

Adherence to quality is paramount when it comes to Controlled Impedance PCBs. So, choose a manufacturer who can produce quality PCBs that meet industry standards. And that adhere to all your specifications.

In this regard, you may want to verify if the manufacturer's quality control practices meet your requirements. For instance, if a company is willing to stand behind their products with an accurate PCB test, then they are worthy. This is because an accurate PCB test will assure you of the quality control that you need when it comes to your controlled impedance boards. These are the aspects that if taken into account, can give you satisfactory results when you decide to work with a Controlled Impedance PCB manufacturer.

References

<https://www.raypcb.com/>

<https://resources.pcb.cadence.com/>

<https://www.tek.com/en>



RAGHAVENDRA ANJANAPPA

System Design-SI,PI & EMI-EMC Engineer

Semiconductor – Package Design/PCB Design/DFx

Applications Engineering – EDA