

Fundamentals of RF and Microwave PCBs

DESIGN GUIDE



SIERRA CIRCUITS

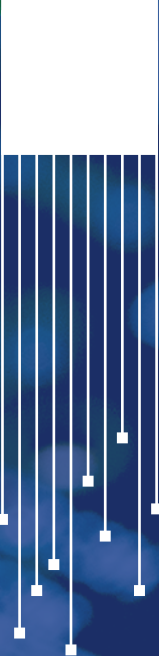
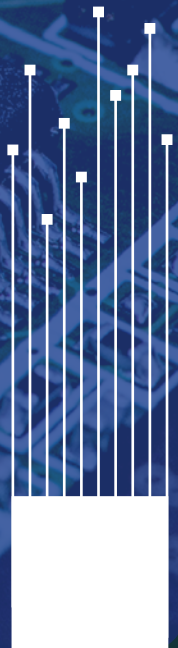


Table of Contents

1. Overview.....	4
1.1. What are RF and microwave PCBs?.....	5
1.2. Basics of RF and microwave board design	6
1.2.1. RF transmission lines	7
1.2.1.1. Microstrip line	7
1.2.1.2. Coplanar waveguide (CPWG)	7
1.2.1.3. Comparison between microstrip line and CPWG	8
1.2.1.4. Stripline	8
1.2.2. What are transmission line parameters?	9
1.2.2.1. Effective dielectric constant (ϵ_r)	9
1.2.2.2. Characteristic impedance	10
1.2.2.2.1. 50 Ω characteristic impedance for RF circuits	11
1.2.3. Impedance matching	12
1.2.3.1. What is a matching circuit?	12
1.2.3.1.1. RF stubs	13
1.2.3.1.2. L network	14
1.2.3.1.3. Matching circuit design aspects	14
1.2.3.2. Return loss and insertion loss	15
2. RF PCB material selection	16
2.1. Important properties of RF board materials	16
2.2. FR4 and modifications	18
2.3. PTFE, ceramics, and hydrocarbons	19
2.4. Rogers PCB material	20
2.4.1. RT/duroid™	20
2.4.2. RO3000™	20
2.4.3. RO4000™	20
2.4.4. Rogers TMM®	21

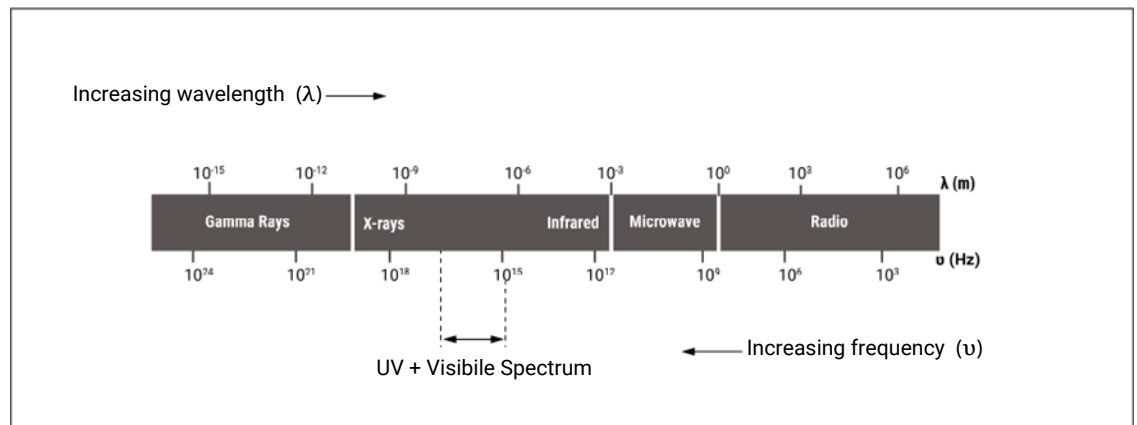
2.5. Materials for bonding	22
2.6. Combination of materials	22
3. Guidelines for RF and microwave PCB design?	23
3.1. What is an RF trace?	23
3.2. Grounding	26
3.2.1. Grounding of shunt components	26
3.2.2. Paddle	26
3.3. Vias	27
3.3.1. Ground stitching vias	28
3.4. Power supply decoupling	29
3.4.1. Decoupling capacitor design guidelines	30
4. RF PCB stack-up	31
4.1. 2-layer stack-up	31
4.2. 4-layer stack-up	32
5. RF component selection and placement	33
5.1. Guidelines for component selection	33
5.2. Design checks for RF component placement	34
6. Testing requirements	35
7. Isolation requirements to avoid interference	36
7.1. Spatial isolation	36
7.1.1. Via fencing	36
7.1.2. X-Y axis isolation	37
7.2. Frequency isolation	38
7.2.1. RF shields	38
7.2.2. Frequency hopping	39
8. RF board applications	41



Overview

The electromagnetic spectrum consists of a number of radiations that are classified based on their frequency or wavelength bands. These include radio waves, infrared waves, microwaves, visible light, ultraviolet rays, and gamma rays.

Notes

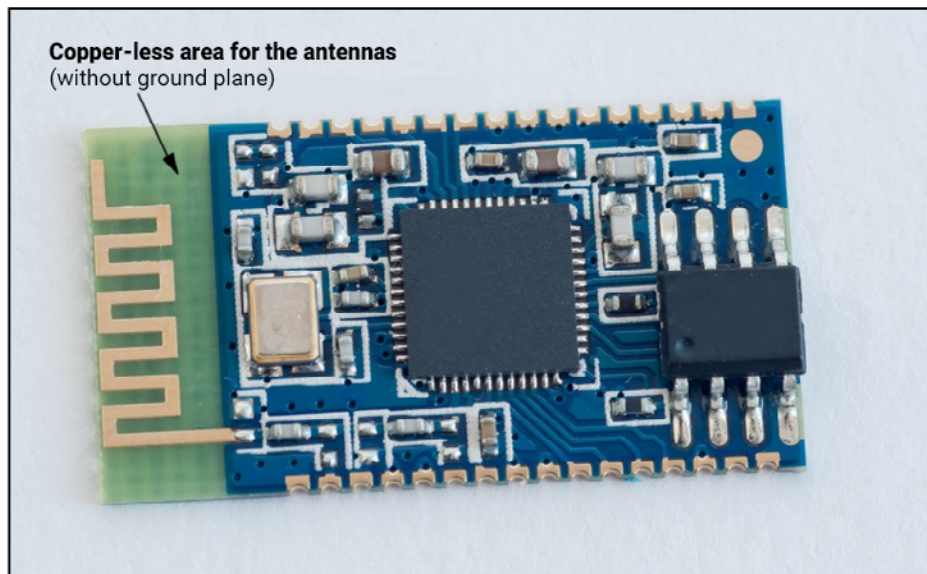


Electromagnetic spectrum

The electronic circuits that operate in the frequency ranges of radio waves (3Hz - 300GHz) and microwaves (more than 1GHz) are considered RF electronics. These devices form a major part of the electronic gadgets available today. We use electronic goods that operate in the high-frequency spectrum. The circuits of these devices are equipped with PCBs designed to provide maximum efficiency while dealing with signals in the radio frequencies.

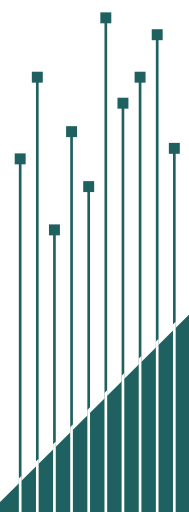
As mentioned earlier, the signals in the electromagnetic spectrum with frequencies between 3Hz and 300GHz fall in the radio frequency range. Among these signals, the ones with frequencies above 1GHz are termed microwaves. The name comes from the small wavelengths that these signals exhibit. These waves are widely used in applications such as radar, cooking, RF transmission, and so on.

1.1. What are RF and microwave PCBs?



A small Bluetooth audio PCB

PCBs that operate at high frequencies to transmit data over the air, using electromagnetic fields are called RF boards. Out of these, the boards that operate above 1GHz are termed microwave boards. These include the Bluetooth® and Wi-Fi modules that are commonly used in all electronic communication devices. Bluetooth modules operate in the 2.4GHz ISM frequency band. This band comprises electromagnetic radiation that falls in 2400 to 2483.5MHz frequency range. Wi-Fi modules work on the standard 2.4GHz and 5GHz frequency spectrums. As more and more devices are preferred to be cord-less, the application of RF and microwave printed circuit boards are increasing.



1.2. Basics of RF and microwave board design

The design of circuit boards meant to operate in the RF and microwave frequency spectrums requires special attention. The parameters such as radio-frequency trace, power supply decoupling, [via holes](#), and [PCB stack-up](#) are particularly designed to work in the radio frequencies. Thickness and cost also increase depending on the application. When it

comes to a wireless device that transmits or receives electromagnetic signals, the main concerns regarding its design are **the antenna and RF layout**. Therefore, the antenna design, the [circuit board layout](#), and the casing of the whole module play vital importance in the overall operation.

During the design phase, PCB designers should ensure there won't be any interference between traces carrying radio waves and the components placed nearby. This type of interference will affect the signal transmission strength of the whole module. Therefore, the layout is arranged in such a way as to reduce the [EMI](#) as much as possible. The image below shows different stages involved in the design and implementation of a radio-frequency or microwave circuit board.



RF PCB design stages

Let us now discuss all the above stages in detail.

Notes

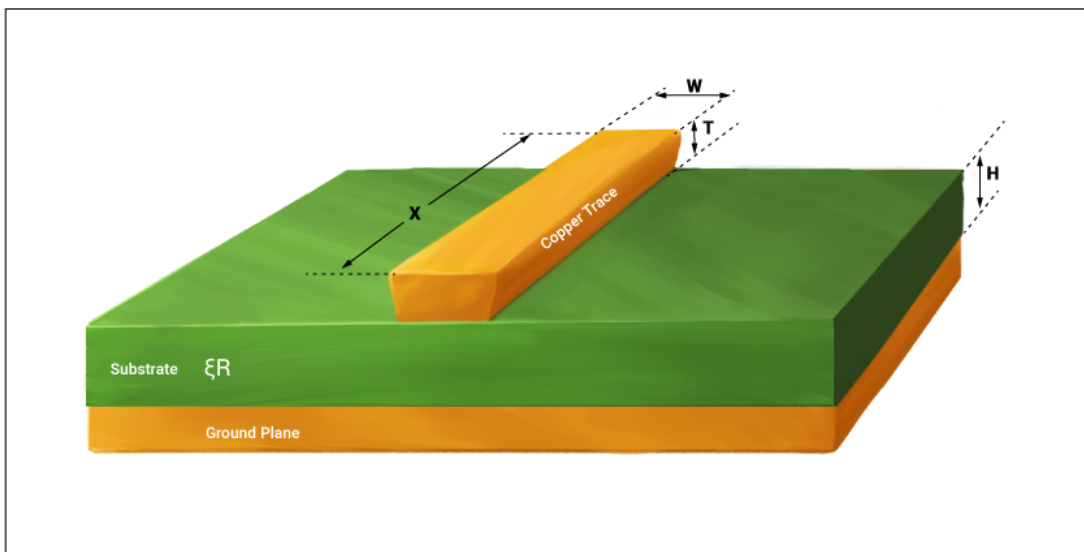
1.2.1. RF transmission lines

[Transmission lines](#) are conductors designed to carry electromagnetic signals from one end to another. The main purpose of these lines is to transmit the signals from the source to the destination with minimal loss.

The transmission lines are classified broadly into the following categories.

1.2.1.1. Microstrip line

A microstrip line is a simple type of transmission line where a signal trace is placed on top of a substrate with a ground plane below. The cross-sectional view of a microstrip line is shown below.



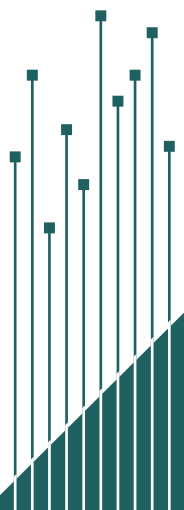
Microstrip transmission line

The important parameters of a microstrip line are the height of the substrate (H), the dielectric constant of the substrate (ϵ_r), trace width (W), trace thickness (T), and trace length (X).

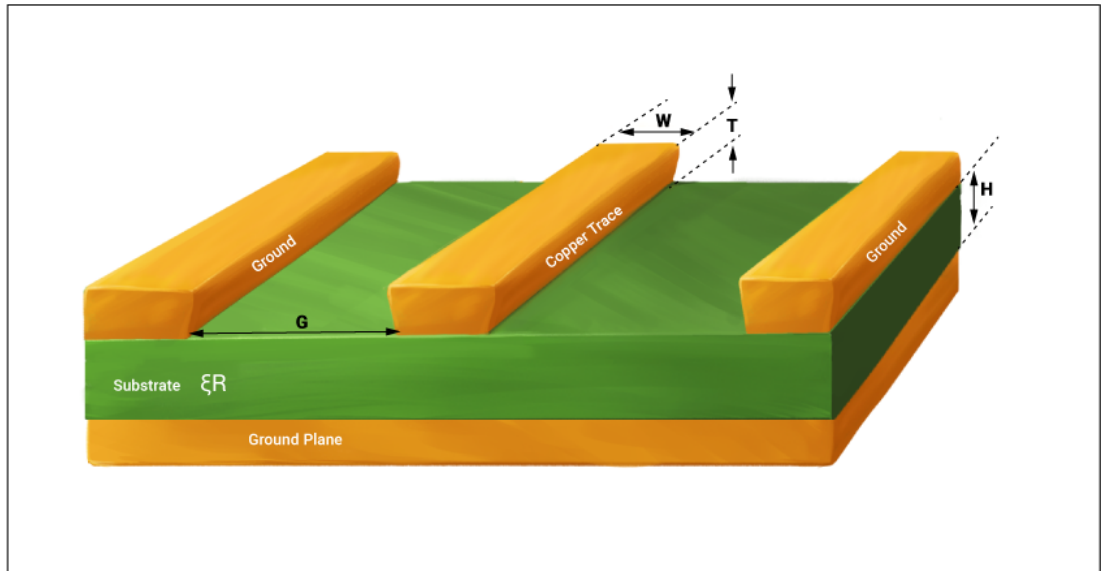
To calculate the required trace widths for your design, check out our [trace width calculator](#).

1.2.1.2. Coplanar waveguide (CPWG)

A coplanar waveguide type transmission line will have a center copper trace with two ground planes on either side and a ground plane below the substrate. These ground planes are kept at a distance of 'G' from



the RF trace as mentioned in the figure below. The other important parameters are the height of the substrate (H), the dielectric constant of the substrate (ϵ_r), trace width (W), trace thickness (T).



CPWG transmission line

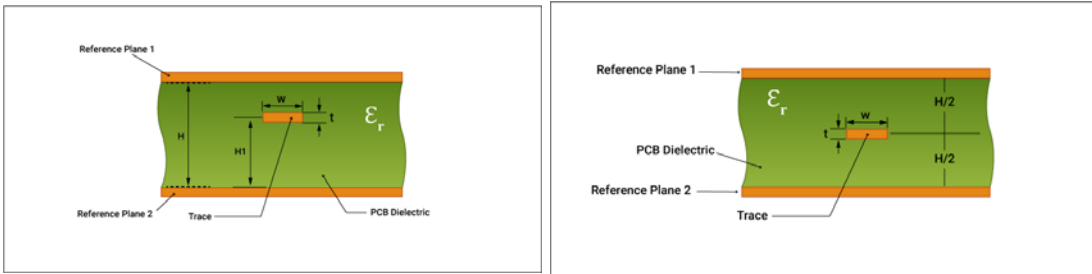
1.2.1.3. Comparison between microstrip line and CPWG

Microstrip line	CPWG
Easy to fabricate	Better isolation and reduced EMI
Effective dielectric constant > CPWG for a given substrate	Minimizes cross-talk
Compact layout	Lower loss at high frequencies

1.2.1.4. Stripline

Stripline is made up of a fixed-width conductor placed in between two reference planes. This means that the trace current has two return paths, either to reference plane 1 or 2. This arrangement offers better shielding to the RF trace and reduces EMI. There are two methods to

place the conductor in this arrangement. One is exactly at the center of the dielectric medium and the other one is offset as shown below.



Stripline cross-section

1.2.2. What are transmission line parameters?

Transmission line parameters are the features that determine the operation and design of any transmission line. At low frequencies and DC voltages, IR losses are more dominant in the transmission lines. IR loss occurs due to the resistance in the transmission line. However, as the frequency increases parasitic capacitance and inductance play a crucial role. The impedance of the trace becomes important and traces behave as transmission lines. The concept of characteristic impedance is essential to understand the transmission of information in transmission lines. Insertion loss, reflection loss, and rise time become relevant.

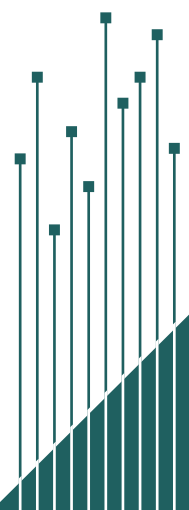
1.2.2.1. Effective dielectric constant (ϵ_z)

In transmission lines, the propagation of signals is considered to be an electromagnetic field propagation rather than a simple electric current movement. Hence, the dielectric constant of air and the substrate, affect the electromagnetic field. The resulting value of both is the effective dielectric constant. The effective dielectric constant of a transmission line is given by the following equation:

$$\epsilon_r = \frac{C_d}{C_{air}}$$

Equation 1

Where C_d is the line capacitance
 C_{air} is the capacitance of the air



1.2.2.2. Characteristic impedance

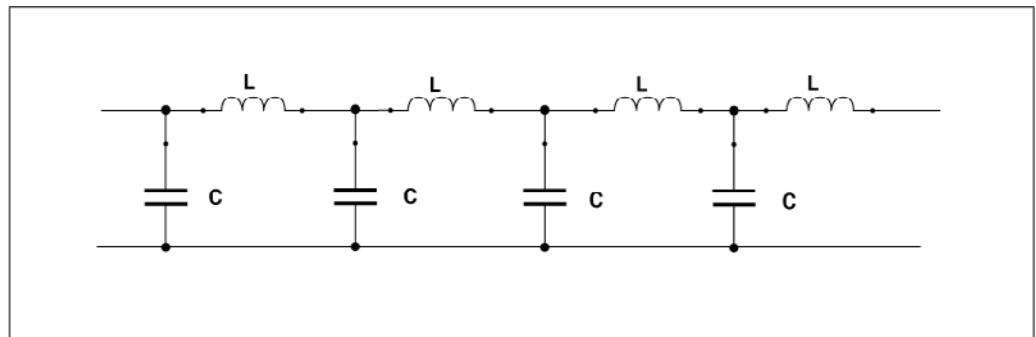
Impedance is a parameter that influences the design of an RF trace or transmission line. When the frequencies are low, the impedance of a transmission line remains the same and does not depend on the distance from the load or trace width. But when the frequencies are high, the impedance changes as the distance from the load or width of the trace varies. Characteristic impedance (Z_0) is defined as the ratio of the amplitudes of voltage and current of a signal transmitting through a lossless transmission line.

$$Z_0 = \sqrt{\frac{L}{C}}$$

Equation 2

Notes

Z_0 is calculated by considering the transmission line as a combination of distributed capacitors and inductors. The capacitors are represented in shunt connections and inductors in series.



Representation of a transmission line for Z_0 calculation

The factors that affect the characteristic impedance of a transmission line are:

- Substrate thickness
- Material type
- Trace width
- Trace thickness
- Distance between trace and ground fill

A general value of **50Ω** is used as a characteristic impedance for most

of the RF circuits. Characteristic impedance helps to calculate the value of the radio-frequency trace impedance. The characteristic impedance at any point in the trace will remain constant, provided the trace parameters are constant. The value of trace impedance Z is calculated using the following equation.

$$Z = Z_0 \frac{(Z_L + jZ_0 \tan \beta l)}{(Z_0 + jZ_L \tan \beta l)}$$

Equation 3

where,

Z : impedance measured at a distance ' l ' from the load

Z_L : load impedance

Z_0 : characteristic impedance of the whole transmission line

β : phase constant

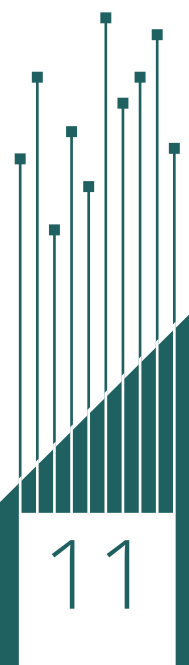
j : reactive part of the impedance

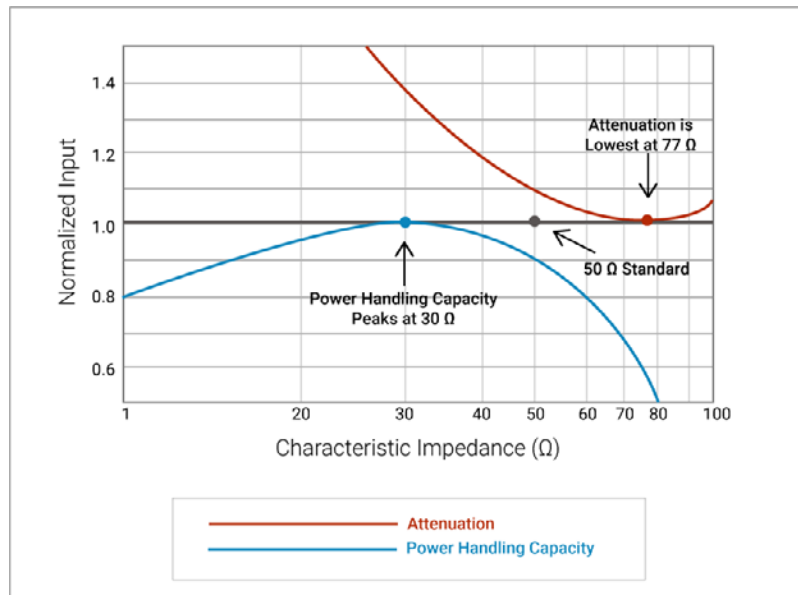
From this equation, it is clear that if $Z_L = Z_0$, then $Z = Z_0$.

Need help to find out the optimal impedance value? Use our [Impedance Calculator tool](#).

1.2.2.2.1. 50Ω characteristic impedance for RF circuits

A characteristic impedance of 50Ω is a trade-off value to achieve minimum loss and maximum power transfer. This can be explained from the graph plotted below.





Normalized input v/s characteristic impedance plot

Notes

In the above chart plotted between characteristic impedance and normalized input values, it is clear that the maximum power transfer between the source and load (transmission) or load and source (reception) occurs at 30Ω . Similarly, the loss is lowest at 80Ω . 50Ω is a value that lies in the middle of maximum power transfer and minimum loss. That is why it is the preferred value of the characteristic impedance in most RF circuits.

1.2.3. Impedance matching

It is clear from equation (3) that the maximum power flow occurs when the impedance of the trace is equal to the characteristic impedance (Z_0). Hence, it is important to match these impedance values so that maximum power transfer happens from the source to load or vice versa. Matching circuits are used to match the measured impedance with the characteristic impedance. Now the measured impedance at a particular point of the RF circuit changes with the respective distances from the source/load. To match this impedance with Z_0 , it is important that it does not change with the distances from source/load. Therefore, matching circuits are required near the source as well as the load.

For example, the standard frequency spectrums like 2.4GHz, assume 50Ω as the characteristic impedance. As a result, impedance matching circuits are designed to match this value.

1.2.3.1. What is a matching circuit?

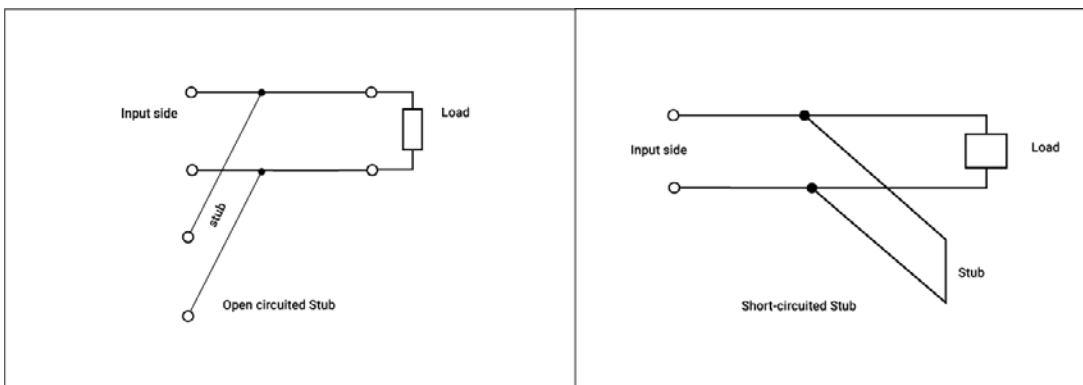
A matching circuit is a passive circuit designed to change the impedance of a circuit to match its characteristic impedance. Reactive components

such as inductors, capacitors, and RF stubs are used as matching circuit components.

1.2.3.1.1. RF stubs

A stub is a portion of a transmission line that is only connected at one end. Stubs are intentionally placed as matching circuits for circuit boards. At times, unintentional stubs are formed on the boards due to specific design patterns. These stubs are able to function as inductors, capacitors, or resonant circuits at radio frequencies. Stubs placed intentionally help to match the impedance of complex loads. Unintentional stubs will increase the attenuation of the whole circuit and should be avoided.

Different types of RF stubs are given below:



Types of stubs

- **Open circuited stub:** The free end of the stub is left open-circuited.
- **Short circuited stub:** The free end of the stub is short-circuited.

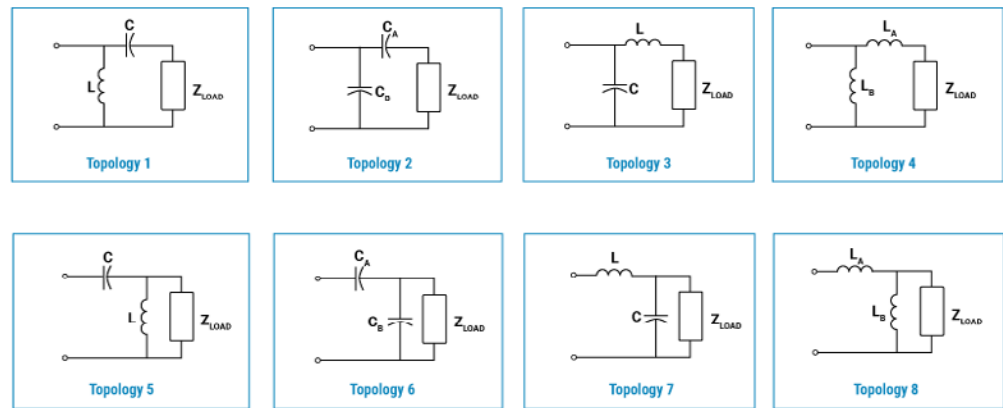
The input impedance of a stub will be either inductive or capacitive and depends on its electrical length and type (open- or short-circuited). As a result, when the operating wavelengths are small, the length of the stubs also reduces. For microwave frequency circuit boards, the size of the stubs is conveniently small due to the low wavelengths.

Matching circuits will have different topologies. It depends on the difference in values between the measured impedance at a point and the characteristic impedance. A capacitance in series, followed by a short coil across the source operates as a simple matching circuit when the difference between these values is minimal. Likewise, different topologies are adopted.



1.2.3.1.2. L network

An L network is a simple lossless circuit that consists of two reactive components arranged in an L formation. Different combinations such as two capacitors, two inductors, one capacitor, and one inductor, etc., are implemented. There are eight standard L network topologies. These are given below:



Standard L-circuit topologies

Two important points to be noted before using L-networks in a circuit are:

- Never use for wideband applications
- Not suitable for circuits operating in gigahertz frequencies

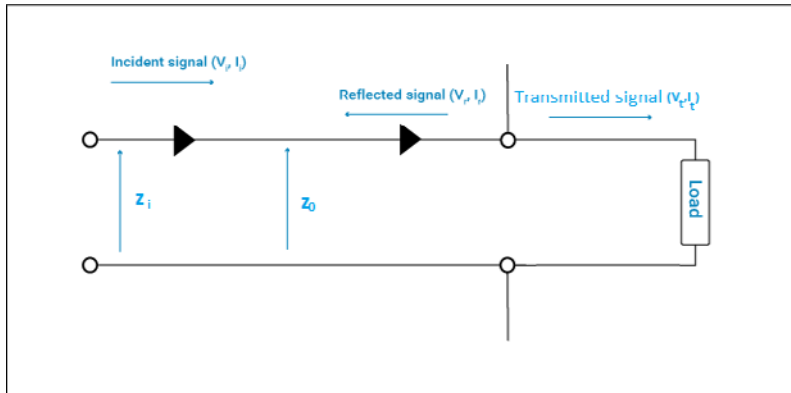
1.2.3.1.3. Matching circuit design aspects

Important aspects to consider while designing an impedance matching circuit:

- Impedance is measured at the exact points where the components are to be placed.
- Avoid using connectors to connect the shunt components. It is better to place the components on the trace itself.
- For capacitors and inductors, choose components with high-Q values.
- For capacitors, series resonant frequency = 2 x operating frequency.
- For inductors, self-resonant frequency = 2 x operating frequency.

Notes

1.2.3.2. Return loss and insertion loss



Different stages of signal transmission through a transmission line

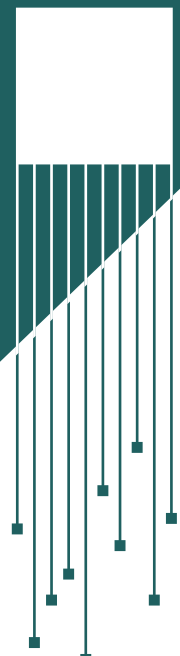
There are certain parameters that determine the effectiveness of matching circuits. When a signal travels from a source to load (incident signal), the initial power it carries is termed incident power (V_iI_i). Ideally, the load should get the complete amount of this power. But in real conditions, only a part of this power is received at the load. The remaining part reflects back to the source since the impedance of the circuit is not exactly equal to the characteristic impedance. The amount of power that goes through is called inserted power or transmitted power (V_tI_t). The rest is called reflected power (V_rI_r). These parameters are used to define the following terms:

- Return loss (dB) = $10 \times \log \left(\frac{\text{incident power}}{\text{reflected power}} \right)$

- Insertion loss (dB) = $10 \times \log \left(\frac{\text{inserted power}}{\text{incident power}} \right)$

- Voltage standing wave ratio (VSWR) = $\frac{V_{\max}}{V_{\min}}$

Among these, VSWR is the most common standard to analyze the efficiency of the transmitted power through a line. Ideally, VSWR should be 1. This indicates that no reflected voltages are there to interfere with the transmitting voltages causing more peaks and valleys in them. Hence the values of V_{max} and V_{min} will be the same.



2. RF PCB material selection

[FR4](#) is the most commonly used material for [PCB fabrication](#). In the case of RF and microwave circuit boards, FR4 causes some limitations. The most prominent one is the change in the dielectric constant values at high frequencies.

The following factors are considered to choose the [best materials for a board](#):

- **Tolerance for signal losses**
- **Stability over high-frequency operating conditions**
- **Impedance stability**
- **Ability to absorb a large amount of heat generated during the operation**
- **Cost of the material and ease of manufacturing**
- **Uniform dielectric constant and loss tangent values over wide frequency ranges**

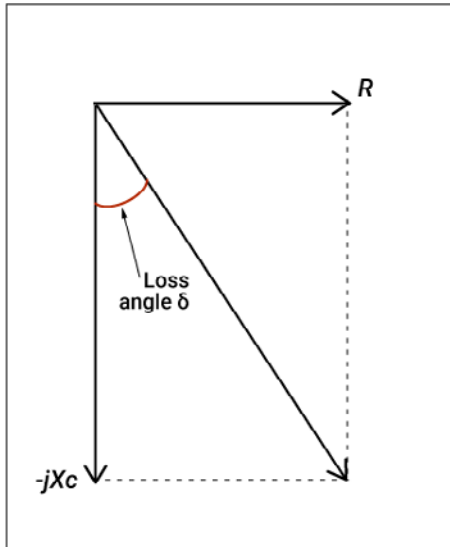
Check out Sierra Circuits [Material Selector tool](#) to find the best substrate for your application.

Notes

2.1. Important properties of RF board materials

- **Dielectric constant (ϵ_r or D_k):** The dielectric constant indicates the ability of a material to store electrical energy when placed in an electric field. It changes with the direction of the material axis. High-frequency operation also affects the dielectric constant. This is the main concern in RF and microwave circuit boards. Therefore, it is important to choose materials that are tested in varying frequency environments. **The materials for radio-frequency PCBs must have a relatively constant ϵ_r value over a wide range of frequencies. Lower the values of ϵ_r , the better, typically in the range of 3 to 3.5.**
- **Loss tangent ($\tan \delta$):** Loss tangent is defined as the loss of signal due to the dissipation of energy at the PCB substrate. This quantity is dimensionless and is given by the phase angle between the resistive and reactive components of a system/component. For example, take

the case of a capacitor. Ideally, it will only have a capacitance C . But in real-life scenarios, it will have an equivalent series resistance (ESR) as well. Hence the loss tangent is illustrated as given below.



Loss tangent illustration

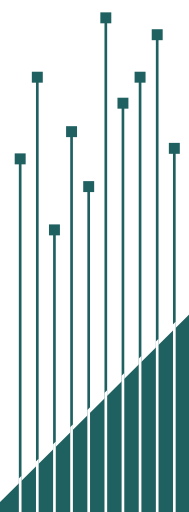
Typical values of tan loss are around **0.0022 to 0.0095** for the frequency range of **10-30GHz**.

- **Coefficient of thermal expansion (CTE):** CTE is the rate at which a material expands or contracts with the application of temperature. It affects various stages of [PCB manufacturing](#) and [assembly](#). It is always better to keep the CTE values of all the materials used in a circuit board stack-up in the same range. If the values are having a lot of differences, it produces thermal stress on the board at high frequencies.

Note to the designer: Choose materials such that there is very little difference between their CTE values for high thermal performance.

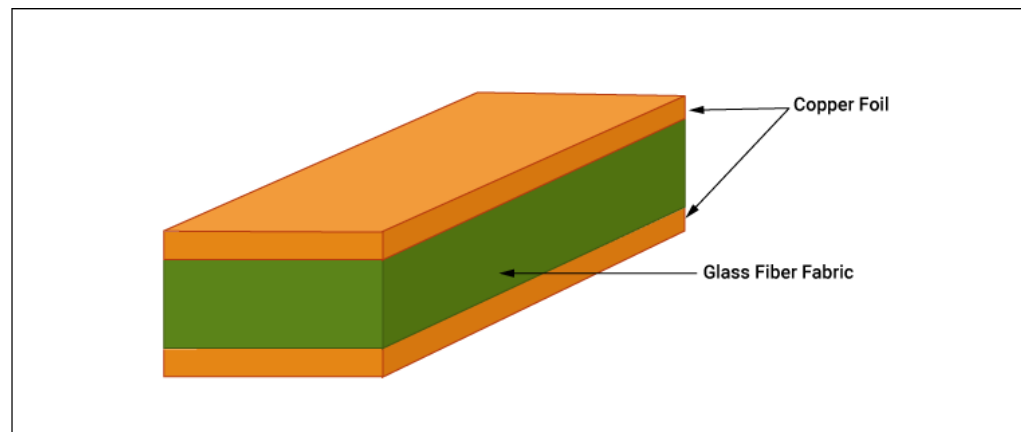
- **Moisture absorption:** The operating environment of the circuit board is considered during the selection of material. Moisture absorption capabilities are not a concern if a PCB is going to operate indoors. But if the operating environment is filled with moisture, then this property of the material is of greater concern.

The commonly used materials for radio-frequency or microwaves are discussed next.



2.2. FR4 and modifications

Plain FR4 (flame retardant level 4) does not perform well under high-frequency conditions. Hence, we consider other options. The selection depends on the operating frequency of the RF board and the incurring losses. For example, suppose a PCB will only operate at frequencies less than 2GHz, and the losses while using FR4 are significant only after 5GHz, then it can be used as the material for the board.



Plain FR4 illustration

FR4 is modified to yield better performance. FR4 derivatives with enhanced ϵ_r values are used for RF boards. When cost is a major concern rather than high-frequency losses, plain FR4 is also used. For example, for normal FR4, FR370HR, the values of dielectric constant and loss tangent are $D_k = 4.04$, and $D_f = 0.021$, and for the FR4 derivative, FR408HR, these are $D_k = 3.68$ and $D_f = 0.0092$.

Thus, if a board operates at high frequencies and the losses are extremely significant, then either FR4 material is enhanced or some other alternative is used.

Notes

2.3. PTFE, ceramics, and hydrocarbons

Most commonly used RF circuit board materials are made up of **Polytetrafluoroethylene (PTFE), ceramics, and hydrocarbons mixed with one another or with a form of glass.**

PTFE with micro-glass fiber/ woven glass	Ceramic filled PTFE	Ceramic filled with hydrocarbon
Great signal quality	Good signal quality	Reduced signal quality
High cost	Reduced cost	Low cost
High CTE	Low CTE	Very low CTE
High rate of moisture absorption	Low rate of moisture absorption	Low rate of moisture absorption
Good electrical characteristics	Good electrical characteristics	Poor electrical characteristics

2.4. Rogers PCB material

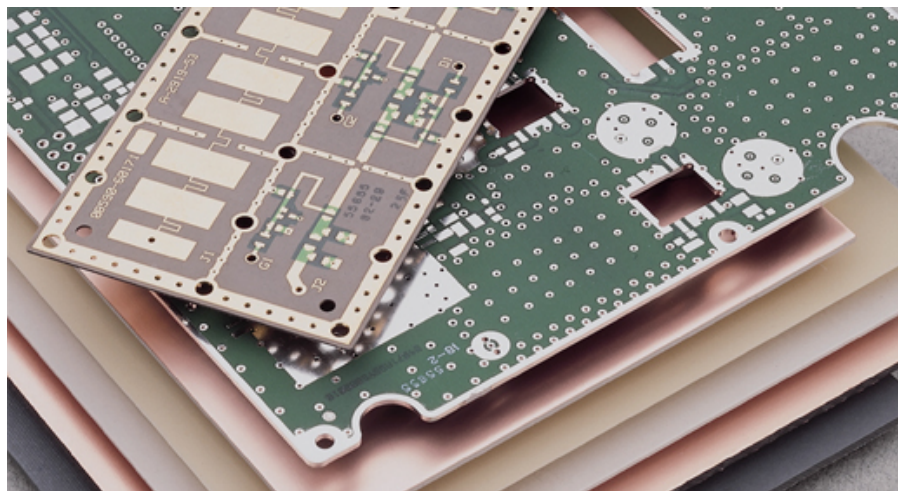
Rogers Advanced Connectivity Solutions (ACS) is a manufacturer of a [PCB substrate material](#) commonly called [Rogers material](#). The core is made up of Teflon or ceramic rather than fiberglass/epoxy. The main advantage of using Rogers material is the high performance provided at high-frequency applications. The table below compares Rogers material with FR4.

Properties	Rogers PCB material	FR4
Cost	High	Lower
Dielectric constant value	In the range of 3 to 3.5	4.5
Temperature management	Less variation with change in temperature	More variation with change in temperature

Dissipation factor	Low	High
Impedance stability	High	Low
Signal loss	Low	High

There are different classes of materials that come under Rogers materials. These are explained below.

2.4.1. RT/duroid™



RT/duroid 5880 laminates. Image credit: Rogers corporation

RT/duroid™ is a composite laminate material with random glass or ceramic-filled PTFE. These substrates are further classified into different types such as RT/duroid 5870, 5880, 5880LZ, 6002, etc. RT/duroid laminates offer reduced electrical loss and moisture absorption, stable dielectric constant, and low [outgassing](#).

2.4.2. RO3000™

RO3000 materials are PTFE filled with ceramic composite laminates suitable for microwave and RF applications. The mechanical properties of these materials are very stable over the high operating frequencies. The dielectric constant values range from 3 to 10.2. These materials are available with or without woven glass reinforcements. RO3003™, RP3003G2™, RO3006™, RO3010™, etc., are some of the variants.

2.4.3. RO4000™

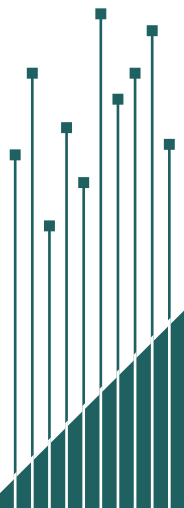


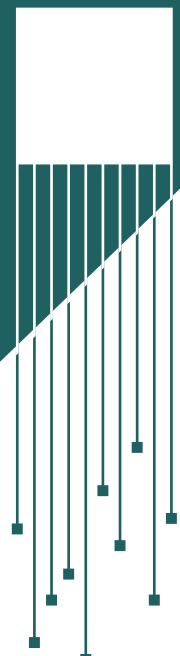
RO4000 LoPro® laminates. Image credit: Rogers corporation

RO4000 laminates are hydrocarbon ceramic materials that are generally used in microwave frequencies. These laminate materials are thermally robust and offer dielectric constant values between 2.55 to 6.15. RO4000 materials are easier to fabricate and are available in UL-94 V-0 flame retardant versions. RO4000 LoPro®, RO4003C, RO4350B, etc., are some of the variants.

2.4.4. Rogers TMM®

Rogers TMM® materials are thermal microwave laminates that come with good mechanical properties, low thermal coefficient, and uniform dielectric constant. These materials are resistant to chemicals and therefore no damages occur during manufacturing and assembly. TMM® 10, TMM® 10i, TMM® 13i, TMM® 3, TMM® 4, etc., are the available variants.





2.5. Materials for bonding

Apart from the materials for laminates, bonding materials are also required to join different layers. These adhesives differ for RF materials as compared to conventional ones. Commonly used ones are fluorinated ethylene propylene (FEP), ceramic-filled PTFE, and liquid crystal polymer (LCP). The important parameters for these materials are low lamination temperature and low re-melt temperature.

2.6. Combination of materials

Notes

Hybrid [multi-layer circuit boards](#) are composite boards made of different materials for different layers. This mix and match of substrates help the PCBs to achieve the desired properties. Therefore, the combination of materials results in properties that are difficult to achieve using a single one. The range of dielectric constant and thermal tolerances are broadened as well.

When the applications are critical and solely performance-oriented, then it is best to opt for hybrid multi-layer boards.

The combination of materials also helps to achieve cost benefits. When the board is manufactured on a strict budget, it is best to use the different materials to reduce the cost. For example, use Rogers material only for the RF layers, and FR4 for the normal component layers. Such construction reduces the material cost.

3. Guidelines for RF and microwave PCB design

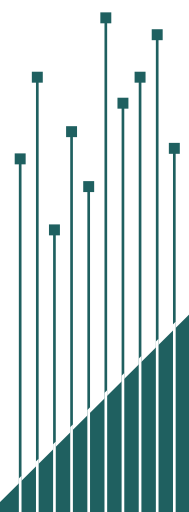
An RF PCB layout differs from a conventional layout in some important parameters. The [EMI interference](#), high-frequency signal channeling, etc., are factors that make the design for a radio-frequency or microwave circuit board complex.

Given below are some basic design guidelines to be followed for an RF board layout:

- The traces are considered as transmission lines and hence either CPWG or microstrip design should be selected.
- The characteristic impedance of the traces should be based on the manufacturer's recommendations.
- Keep a uniform trace width to achieve a constant characteristic impedance value.
- In the case of CPWG traces, the gap between the RF trace and ground planes on either side should be constant throughout.
- Make sure that in the top layer of a CPWG transmission line, the gap between the ground planes is less than the height of the substrate.
- The ground impedance is kept as low as possible. High-frequency circuits require minimum impedance. When an RF IC component is grounded through a single via or a narrow trace, the resulting ground impedance value is substantial. Hence, it is best to use more than one via for grounding. A pair of vias instead of one will reduce the resulting ground impedance to 50%. Also, add a ground plane under an IC or an RF trace. All these design techniques reduce the ground impedance values.
- Use ICs wherever possible as components. These ICs are already tested by the manufacturer and come in compact packages.

3.1. What is an RF trace?

An RF trace is a trace on a PCB layout designed to propagate radio frequency signals with minimum losses. Apart from the guidelines given

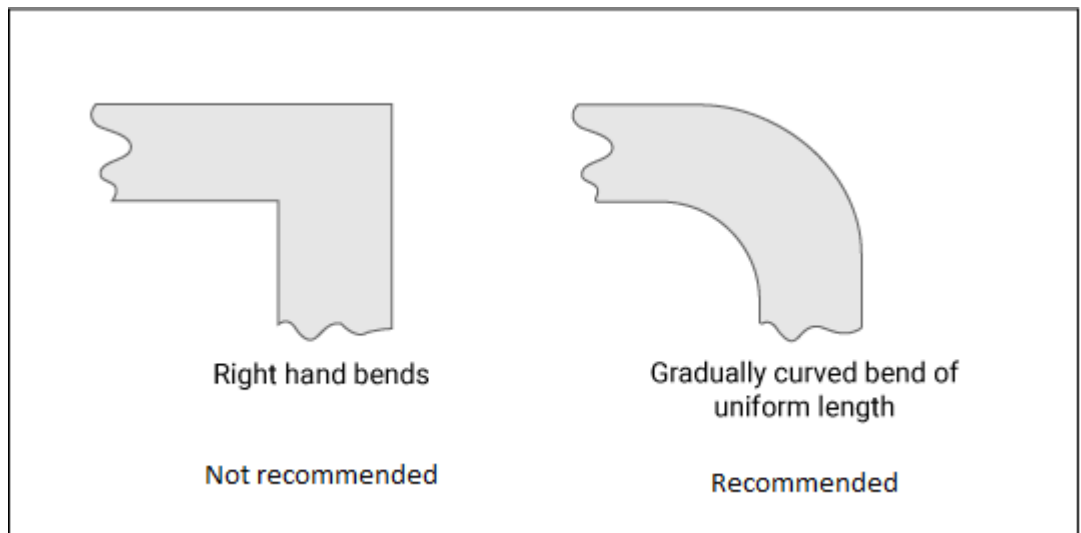


above, there are specific design rules for RF traces. These traces are the transmission lines whose characteristic impedance decides the overall performance of the circuit board.

Let us have a look at the trace requirements:

- Keep the trace lengths as short as possible. As the trace length increases, the attenuation also increases.
- Never place a normal trace and RF trace parallel to each other. This results in interference of signals.
- Always provide a ground plane for establishing the return path of the trace currents.
- Test points should not be placed on the traces. They will disturb the matched impedance values.
- Avoid bends in the trace. Bends tend to change the controlled impedance values of an RF trace. But in situations where bends are unavoidable, try to make a uniform gradually curved bend rather than a sharp right-angled one as shown below.

Notes

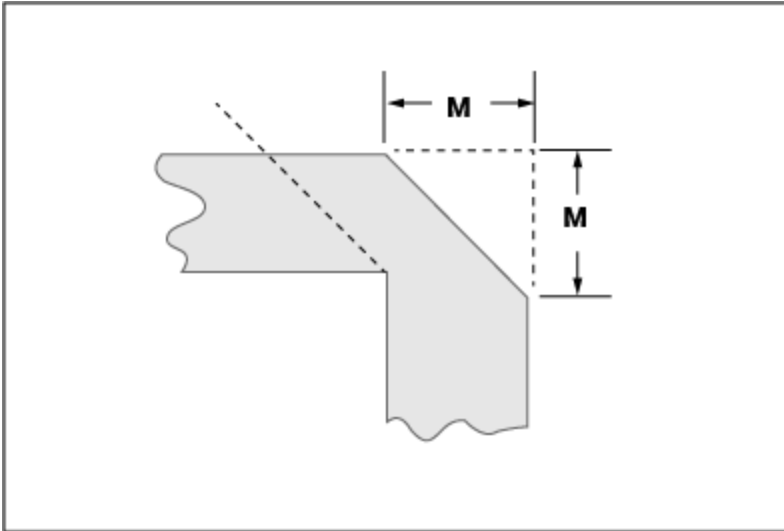


Desirable bends in RF traces

- The bend radius needs to be of a value as given in the following equation:

$$\text{Bend Radius} \geq 3 \times (\text{Line Width})$$

- If a right-angled bend is absolutely necessary, then a corner compensation method, as shown below should be employed



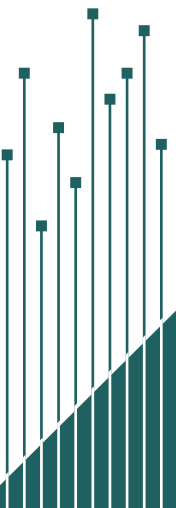
Mitering of an RF trace

This method is called mitering of the trace. The amount of material to be cut out for mitering is calculated by using the formula.

$$M = W \left(1.04 + 1.3 * e^{-1.35 \frac{W}{H}} \right)$$

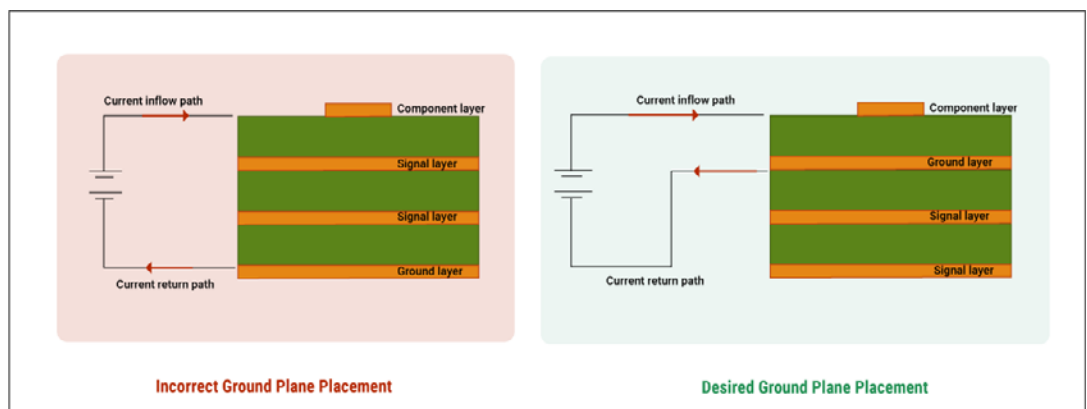
Where M: width of mitering
W: width of the RF trace
H: height of the substrate

- All the RF traces require proper grounding planes to establish a return path for the signals.
- Use stubs only if absolutely necessary. A stub will increase the reactive part of the impedance. Thus, it is better to place the components on the traces rather than introducing a stub to connect them to a trace.



3.2. Grounding

Grounding ensures a return path for the current propagating through a trace or component. In [PCB design](#), ground planes are assigned to carry out this process. In RF boards, there are some special considerations for the ground planes. For every layer with radio-frequency traces and components, a dedicated ground plane is required. This plane should be placed immediately under the component plane to make the return path the shortest.



Desired ground plane placement

The ground plane should never be broken. If this plane is broken or interrupted, the return currents will flow through some other short path. As a result, a new current loop is formed that creates unwanted inductance in the circuit. The ground plane is designed as wide as the design allows it to be. Narrow planes will increase signal leakage. It is a good practice to use ground planes above and below a conductor to improve the performance.

3.2.1. Grounding of shunt components

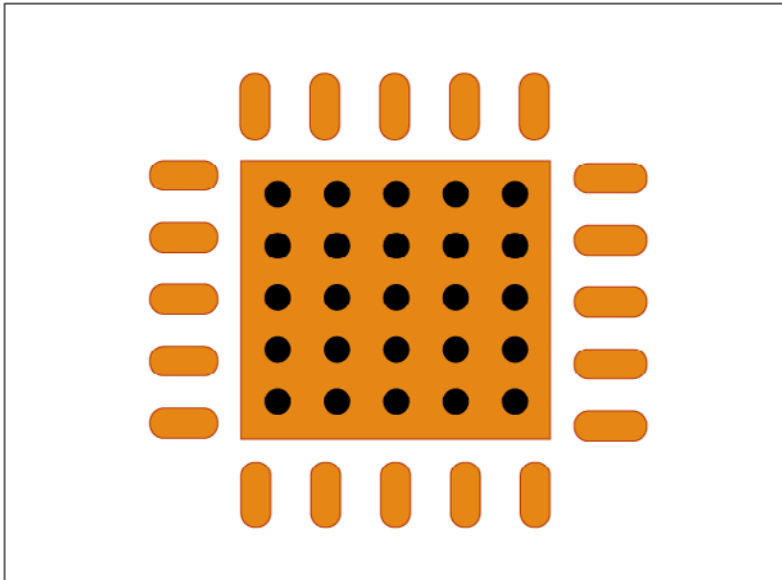
The components connected in parallel to the RF trace have a peculiar grounding requirement. **A minimum of two grounding vias is needed for any shunt component in an RF transmission line.** More than one grounding via, as explained earlier, reduces the parasitic capacitance.

3.2.2. Paddle

A ground paddle is a solid ground plane provided directly under an IC component on the component layer. Apart from providing a return path for the current, paddles act as heat sinks. Therefore, the maximum possible

Notes

number of vias as per the layout considerations are accommodated in these paddles.



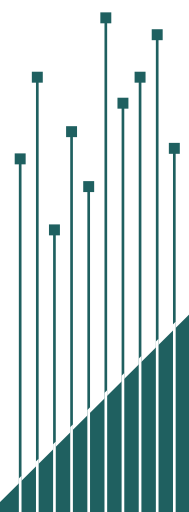
A ground paddle with vias under an IC component

The vias implemented for the heat transfer are ideally plated through-hole vias filled with thermally conductive paste to assist the process.

3.3. Vias

Vias are used to interconnect different layers in the PCB. The introduction of a via induces parasitic capacitance and affects the high-frequency operation of RF boards. Hence, in these boards, the design and implementation of vias need to follow some additional guidelines:

- Dedicated vias for each pin or pad of a component.
- The vias between the top layer ground fillings and inner ground layer should be:
 - Spaced at a distance, not more than $1/20$ of the signal wavelength.
 - As many as possible as allowed by the design.



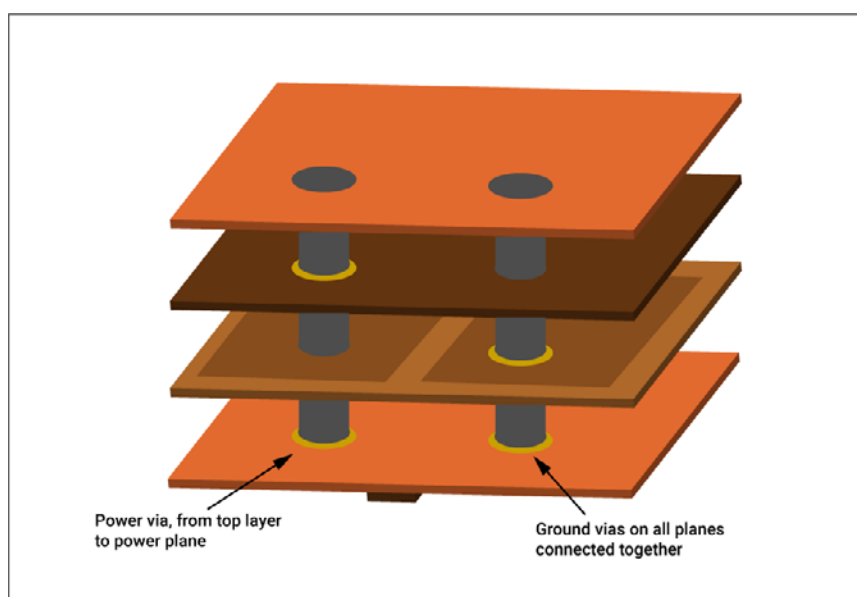
Frequency	Free-space wavelength (λ_0)	Wavelength in FR4 ($\epsilon_r = 4$)	$1/20 \lambda$ (FR4)
2.5 GHz	12 cm (120 mm)	6 cm	3.0 mm
3.0 GHz	10 cm (100 mm)	5 cm	2.5 mm
4.0 GHz	7.5 cm (75 mm)	3.75 cm	1.88 mm
5.0 GHz	6 cm (60 mm)	3 cm	1.5 mm

- Restrict the use of vias to route an RF trace from a layer to another.
- Ground vias are required near all the pins in the top layer.
- Introduce more parallel vias to reduce the parasitic inductance.

3.3.1. Ground stitching vias

Notes

Via stitching is a process to establish connections between copper areas in different PCB layers using vias. Ground plane via stitching ensures a shorter ground return path for the current. This low resistance path reduces heat dissipation and voltage drop. If it is not done, then different ground planes will have different resistances and voltage drops. Thus ground plane via stitching uses vias to connect all the ground planes and maintains a uniform voltage drop throughout the circuit board.

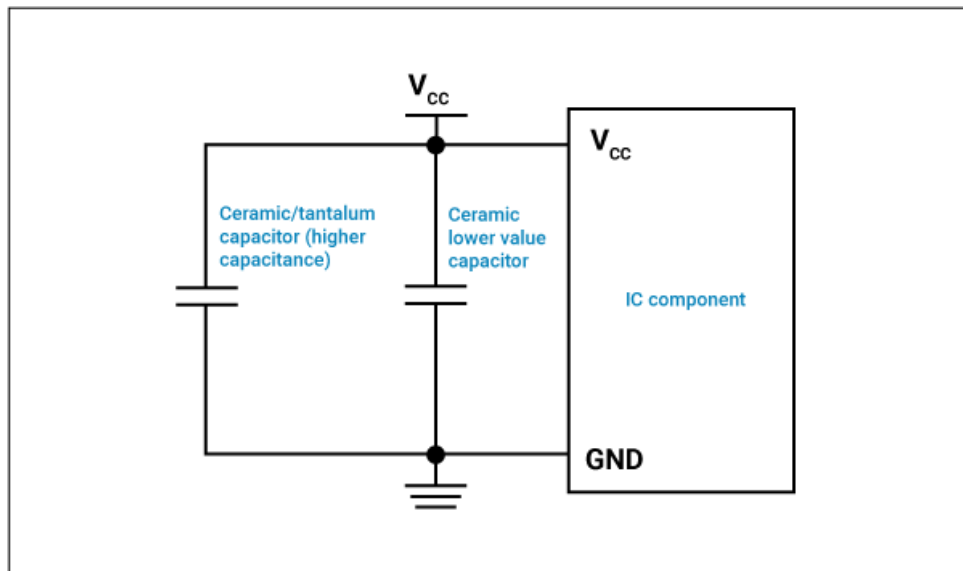


Ground vias connected together

3.4. Power supply decoupling

Power supply decoupling is the process of filtering out the noise from the power supply signals. Capacitors are used for this purpose and these capacitors are termed decoupling capacitors. Noise that is carried on from the power supply will affect the quality of the signal. It also affects the sensitivity of the circuit, causes undesired interference, and thus reduces the RF output quality. Therefore, decoupling capacitors are connected across the power supply.

Decoupling capacitors increase the impedance of the existing circuit. Hence, these capacitors should have the least impedance in order to maintain the impedance matching of the RF circuit. A capacitor will offer minimum impedance if it is operated at the self-resonant frequency (SRF). Therefore, first, the frequency of the noise that occurs in the circuit is calculated, then capacitors with SRF closer to that frequency are chosen.



Power supply decoupling capacitors

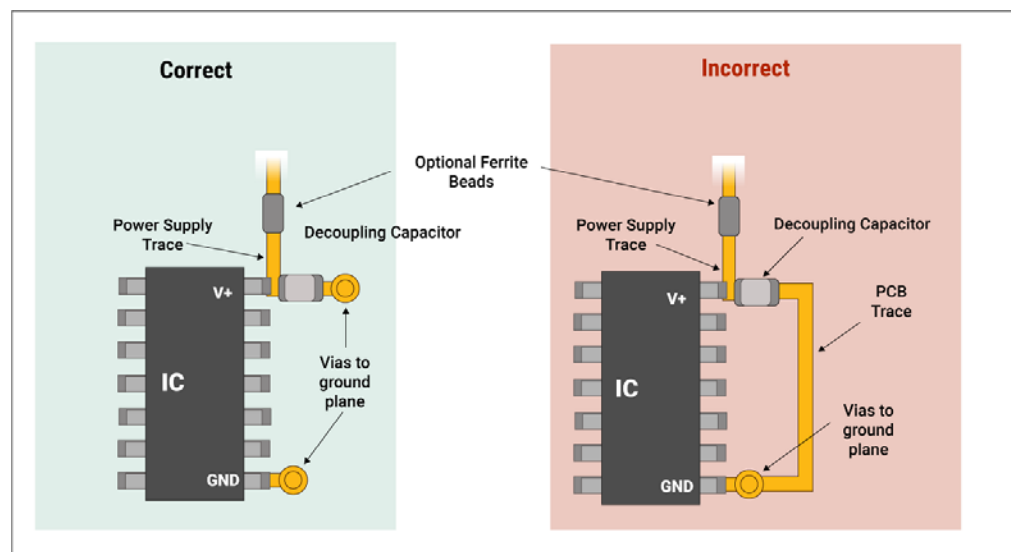
For a particular IC package, the SRF value of a capacitor will be higher if the capacitance value is lower and vice versa. The figure above shows a simple circuit with two decoupling capacitors connected in parallel. The capacitance value of one is higher and the other is lower. The higher capacitor filters the low-frequency noise in the system and stores the energy. The lower capacitor filters out the high-frequency noise.

3.4.1. Decoupling capacitor design guidelines

The following are the design considerations for placing decoupling capacitors in radio-frequency circuit boards:

- Place the decoupling capacitors on the same layer with the components.
- The capacitor with the lowest value of capacitance should be placed near the power supply.
- Place the capacitors as close to the IC pins as possible.
- Use small capacitance for high-frequency transients and large capacitance for low-frequency transients.
- Always place the capacitors in parallel to the signal path.
- Each decoupling capacitor requires a dedicated ground via. Never share ground vias.

Notes



Decoupling capacitor placement guidelines

4. RF PCB stack-up

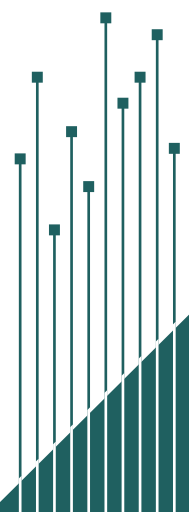
The design for any PCB stack-up has the main agenda to preserve the [signal integrity](#) and [power integrity](#) as much as possible. In the case of RF boards, it is critical to keep maximum power and signal integrity. The three most important aspects in the radio-frequency stack-up design are given below:

1. Isolation between components and traces: The chances of interference between signals propagating through nearby circuit elements are high in RF circuit boards. This is because of the high-frequency operation. Isolation between circuit elements in the same layer is taken care of during the layout design of the board. But the same problem arises between layers as well. Hence, during the stack-up design, this issue is considered and enough isolation is provided.
2. Decoupling: Proper decoupling is required between ground and power planes.
3. Number of layers and their arrangement: Number of layers required to give the proper isolation, decoupling, and their arrangement, is of utmost importance. Power and ground planes are placed as adjacent internal layers. Components are placed on the surface layer nearest to the ground plane.

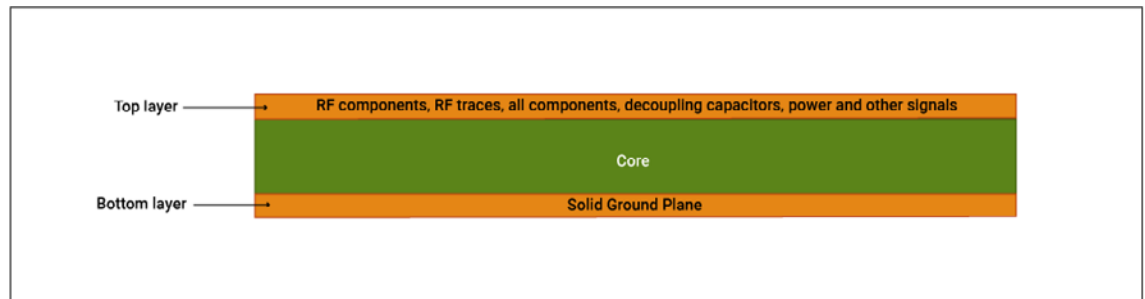
To get an optimized stack-up, try our [Stackup Designer tool](#).

4.1. 2-layer stack-up

2-layer stack-ups are generally not preferred for RF circuit boards. This stack-up will not be enough to give the required clearance to deal with high-frequency boards. But in some applications, to save cost and make the design simple, a 2-layer stack-up is employed. The operating frequencies of these PCBs will be in the kHz range and not in the GHz range. The room available to route the radio-frequency components separately will be very less in this design. The designers should try to include all the RF components, decoupling capacitors, traces, etc., in the top layer and try to provide an individual layer for the ground. An



example stack-up is shown below:

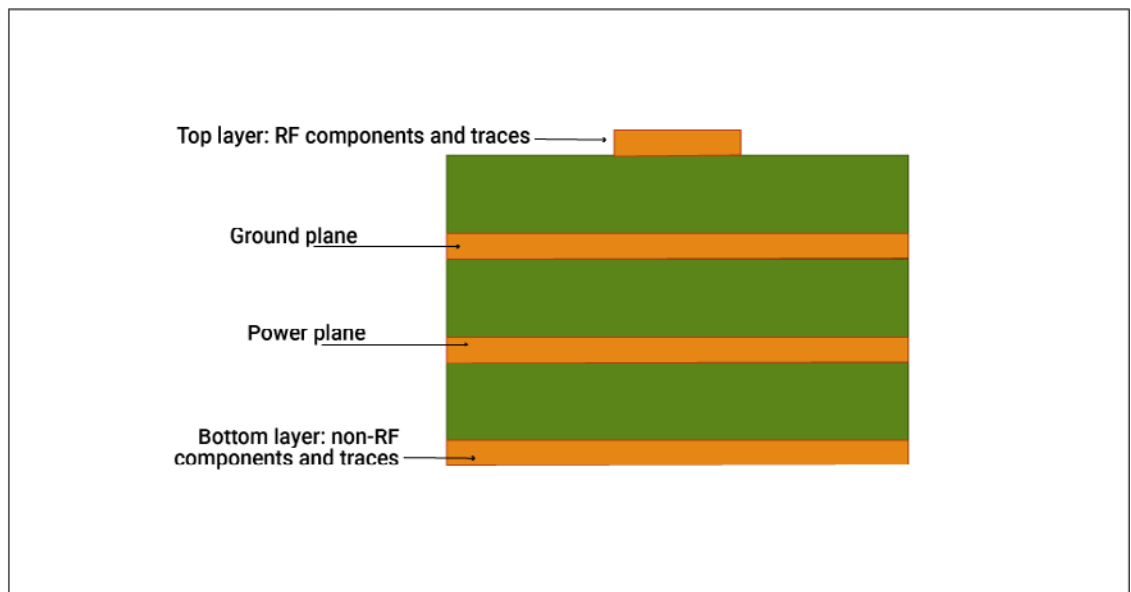


A typical 2-layer RF PCB stack-up

Notes

4.2. 4-layer stack-up

4-layer stack-ups are the most commonly used structure for RF PCBs. This stack-up offers a dedicated ground and power plane. All the radio-frequency signals and components are arranged in a single layer. Power and ground planes are arranged as inner layers adjacent to these component layers. The substrate thickness between an RF trace and the grounding plane also plays a role to determine the trace thickness. Hence, to design a stack-up, it is necessary to consider this fact. An example 4-layer stack-up is given below:



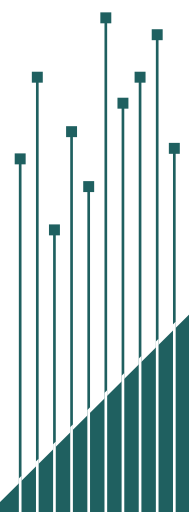
4-layer RF stack-up example

5. RF component selection and placement

The RF components that are mounted on a board are always carefully selected in order to enhance the operation. The basis of component selection is the high-frequency tolerance and operation. But apart from this, there are several guidelines to be followed.

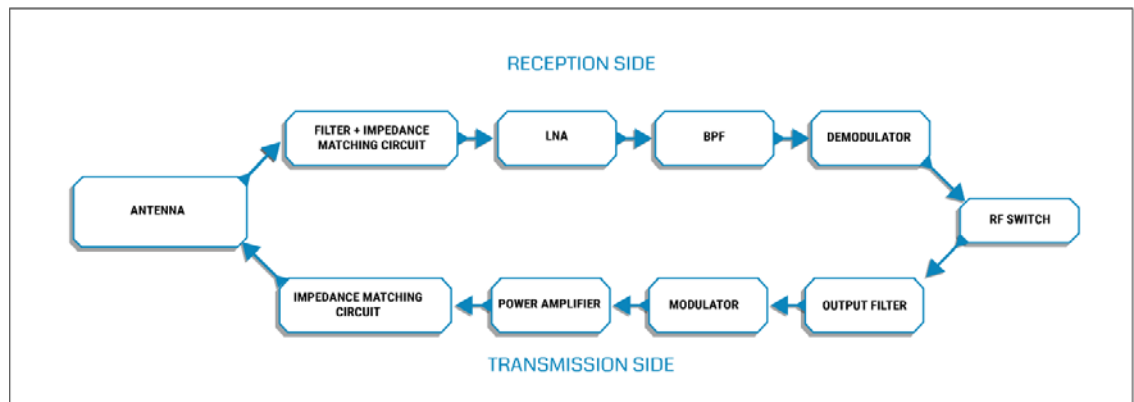
5.1. Guidelines for component selection

- Capacitors made of ceramic COG/NPO materials are preferred (for matching circuits). These types of capacitors are stable over a wide range of temperatures.
- X5R or X7R capacitors are suitable for decoupling.
- Capacitors with low ESR values are chosen.
- Quality factor (Q) of the capacitors should be high. It is the ratio between the reactance and resistance of the capacitor at a given frequency.
- The recommended package sizes are miniature ones like 0402, 0201, etc.
- Inductors with high Q values are used for matching networks.
- The SRF for matching circuit inductors should be above the operating frequency.
- Choose wire-wound ceramic inductors.
- The component bandwidth should be such that it matches the required gain, power output, and expected power consumption.



5.2. Design checks for RF component placement

The first rule of [component placement](#) for any PCB is to try and follow the signal flow path. This remains the same for an RF circuit board as well. The path for both the transmission and reception sides is illustrated in the figure below.



Signal flow path for an RF communication circuit

Other than this, the following are the checks to carry out before placing RF components on a high-frequency board.

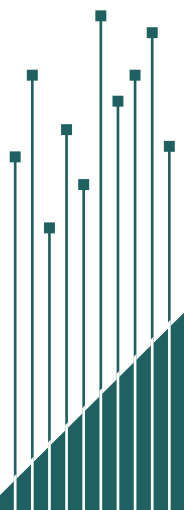
- Never place the inductors on the radio-frequency circuit parallel to each other with very little distance in between. This is to avoid cross-talk caused by the mutual inductance between them. It is best to place them orthogonal to each other.
- The antenna requires a keep-out area around it. This is the area within which no other metallic objects should be placed. The keep-out area avoids the placement of components in the near field around the antenna that might affect its performance. It is not specific to components alone, other board parts such as mounting screws also shouldn't be there in this specific region.
- Use a low-noise amplifier (LNA) at the reception side to reduce noises to a minimum.
- Give enough clearance between the transmission and reception channels.

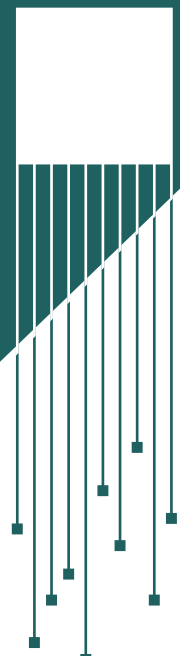
Notes

6. Testing requirements

Conventional circuit boards are tested by placing test points on the circuit. But for an RF board, placing test points will increase the impedance and stub lengths of the PCB. Therefore, tests such as in-circuit, [flying probe test](#), etc., cannot be carried out on a radio-frequency printed circuit board.

- All the tests involving the radiation around the circuit board are done in a controlled environment. This can be a sterile room or a faraday cage.
- Connectors are provided for testing purposes alone. These are removable after testing and may not be part of the final product. These connectors verify the matching circuitry.
- Bluetooth low energy (BLE) testers are used to check the functionality of RF paths and to record the parameters.





7. Isolation requirements to avoid interference

Isolation is an extremely important factor to consider for high-frequency PCBs. Since the frequencies are always in the radio-frequency or microwave range, there are fields created around the conducting traces and components. The interaction of any sort leads to losses of the signal. Hence isolation requirements are properly studied during the design of RF boards. The three types of isolation are discussed below.

Notes

7.1. Spatial isolation

Spatial isolation helps to provide the required physical separation between components or traces. Antennas are the elements in an RF circuit for which spatial isolation is critical. In most modern-day high-frequency boards, Wi-Fi and Bluetooth antennas are present. Both these antennas need maximum possible separation from each other. At the same time, the total available space is also limited. Therefore, different techniques are used to provide the isolation. Some of these techniques to obtain spatial isolation are discussed below:

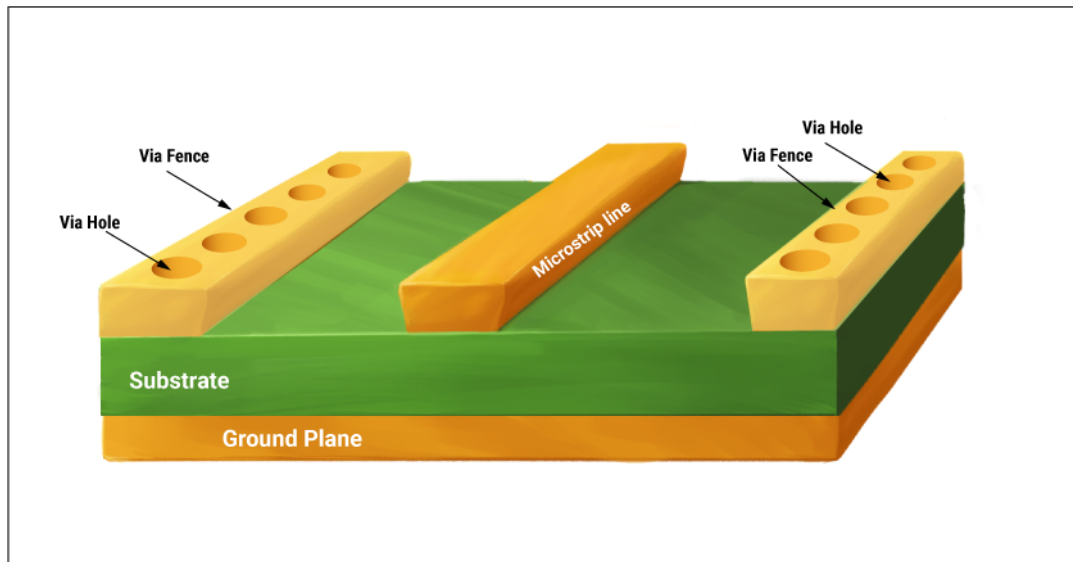
7.1.1. Via fencing

One of the most popular techniques for isolation is called via fencing. This method is generally used for the following:

- Shield microstrip and stripline transmission lines
- Provide fencing around circuit board edges
- Separate different functional units on the same board

This is a cost-effective and simple method to provide spatial isolation. It is used in contemporary circuit boards for BLE and Wi-Fi unit isolation. This technique also prevents parasitic coupling between traces laid out in close proximity to each other.

A via fence is formed by implementing a row of vias through the substrate. In stripline transmission lines, via fences are employed on both substrates above and below the RF trace. Thus grounding of these vias is done using the respective top and bottom ground planes. But in the case of microstrip lines, the top side does not have a ground plane. Hence, the via fencing in the top layer is connected together with a metal track as shown in the figure below.

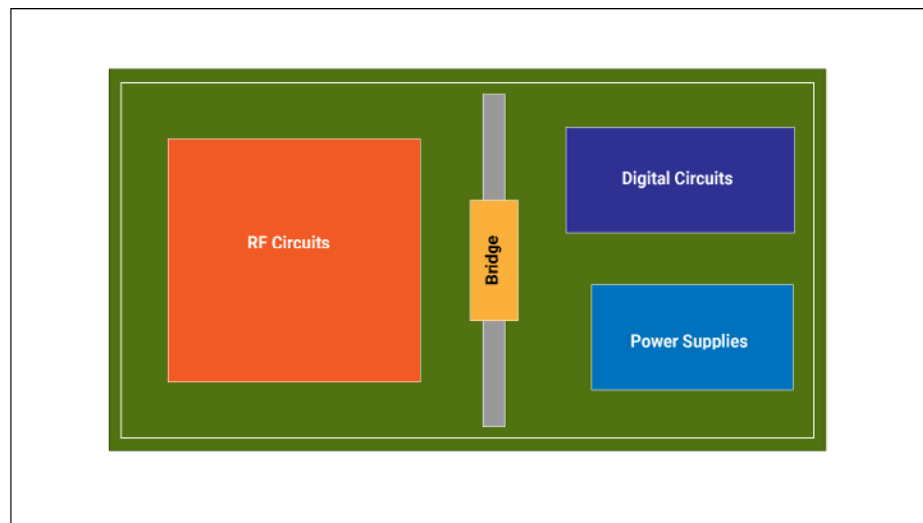


Via-fencing in microstrip lines

7.1.2. X-Y axis isolation

Most of the RF circuit boards come with a mixed-signal design. Therefore, different circuit types on the boards need maximum isolation from each other. X-Y axis isolation is an easy way to achieve this. The major demerit of using this type of isolation is that it requires a slot in the stack-up. This slot will thus interrupt the continuity of the ground plane, which is not desired. To avoid this, a bridge area is provided to route the circuitry over from one part of the board to another. This

ensures the continuity of the planes and circuits.
The floorplan of such isolation is shown below:



Floorplan for X-Y isolation

Some of the important design aspects for X-Y isolation are mentioned next:

1. The slot may be a physical cut-out or a copper layer slot
2. RF layer and digital layer grounds are connected through the bridge
3. The bridge is formed on a ground plane at a specific location
4. The slot is designed to be a no-go zone with an exception only for the ground layers since they need to be continuous
5. This method of spatial isolation is effective when the digital signals do not penetrate much into the RF zones of the PCB
6. It helps to divide the digital and analog routing into two separate zones

7.2. Frequency isolation

Isolation in frequency is achieved using either of the two methods listed below:

7.2.1. RF shields

RF shields are conductive or magnetic material barriers to block unwanted radio-frequency electromagnetic signals. These prevent frequency interference between nearby components and traces. It becomes crucial in the case of high-frequency components.

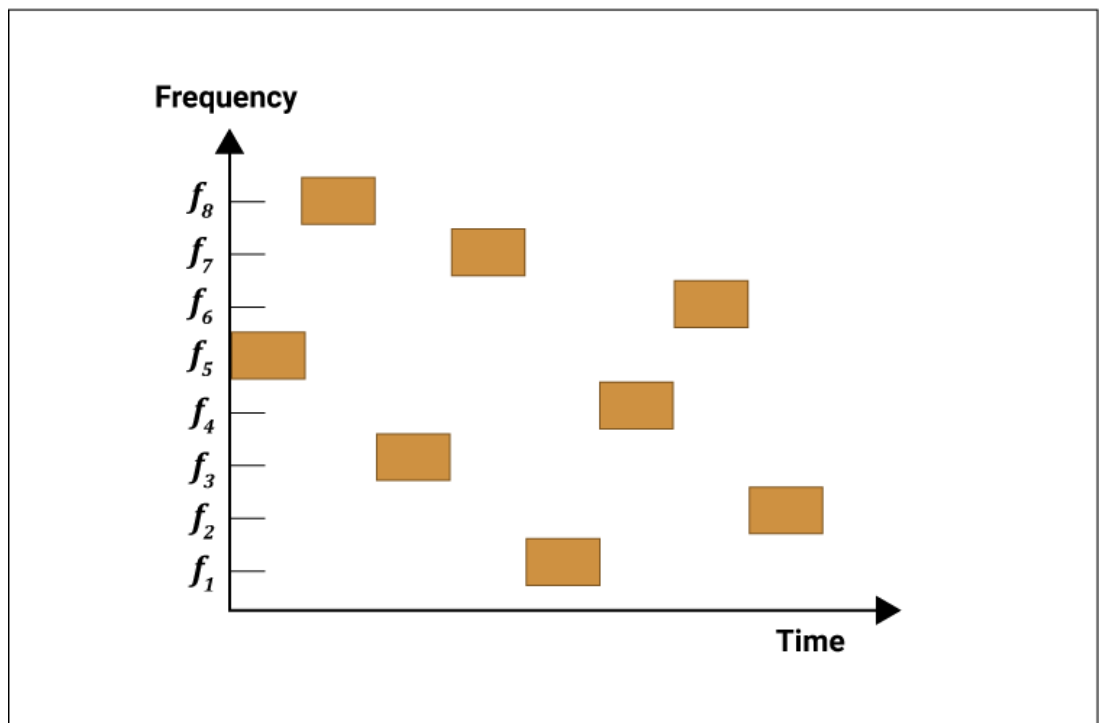


RF shields for isolation

7.2.2. Frequency hopping

Frequency hopping is a technique used to prevent signals from interfering with each other. The RF signal transmitters change the frequency of the carrier wave rapidly to occupy a wide spectral band. A code is employed for this purpose that is known to both the transmitting and receiving devices. This process is called frequency-hopping spread spectrum (FHSS).

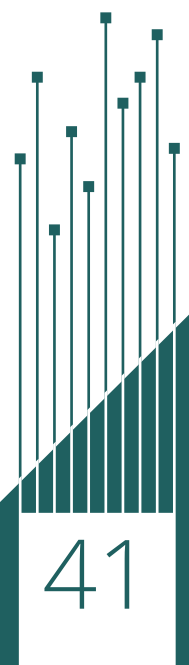
Notes



A plot representing frequency hopping

The frequency band is further divided into many sub-bands and the carrier wave frequency is switched between these sub-bands. Usually, the Bluetooth and Wi-Fi modules are located closer to each other. Both of these technologies are prone to failure due to interference. Therefore, it is important to provide isolation between these modules, since both are operating in identical frequency spectrums.

Wi-Fi uses a direct sequence spread spectrum (DSSS) and the carrier wave does not change the frequency. In comparison, Bluetooth uses FHSS and hops between 79 channels that are 1MHz wide. Thus frequency isolation is attained between components in close proximity by frequency hopping.



8. RF board applications

RF boards are required in a wide range of modern-day devices that include smartphones, tablets, laptops, wireless technology, sensors, actuators, robotics, and many more. It is not possible to list out all the applications that require the involvement of these circuit boards. The materials that are used for the fabrication of radio-frequency and microwave PCBs are application-specific. The chart below shows the type of materials recommended for different areas of application.

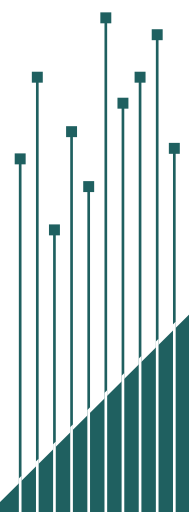
Notes

Application	RF material
Space and military	RT/duroid RO4000
Medical	RO4350B
Consumer electronics	RO3006 RO3010 RO4835
Automotive	RO3003 RO4000 RO4350B
High power applications	6035HTC XT/Duroid
Industrial	RO4835 RO4350B XT/duroid

The key takeaways from this design guide are:

1. RF and microwave PCBs form the basic building blocks of the wireless connectivity designs widely in use today.
2. The RF traces are equivalent to transmission lines carrying high-frequency signals.
3. These radio-frequency transmission lines are impedance matched to retain a 50Ω characteristic impedance throughout.
4. Other critical design aspects of these circuit boards include trace parameters, layout, power supply decoupling, ground layer continuity, and stack-up.
5. Proper isolation is given to the RF traces and components to prevent interference.
6. The type of substrate material decides the application of the board.

All of the aspects discussed throughout this guide are in one way or another important in the process of a radio-frequency or microwave PCB design. We have tried to include all the solutions to the problems one might face during this process. Please get in touch with us to solve your queries or to place an order at protoexpress.com.



Sierra Circuits, Inc.
1108 West Evelyn Avenue
Sunnyvale, CA 94086

We provide our customers with unprecedented quality, reliability,
and a single point of support. No more miscommunication
between multiple vendors and no more delays.

We are **ISO-9001:2008**, **ISO 13485:2016** and
MilSpec MIL-P-55110 certified.

