Radio - Frequency Printed Circuit Boards (RF - PCBs)

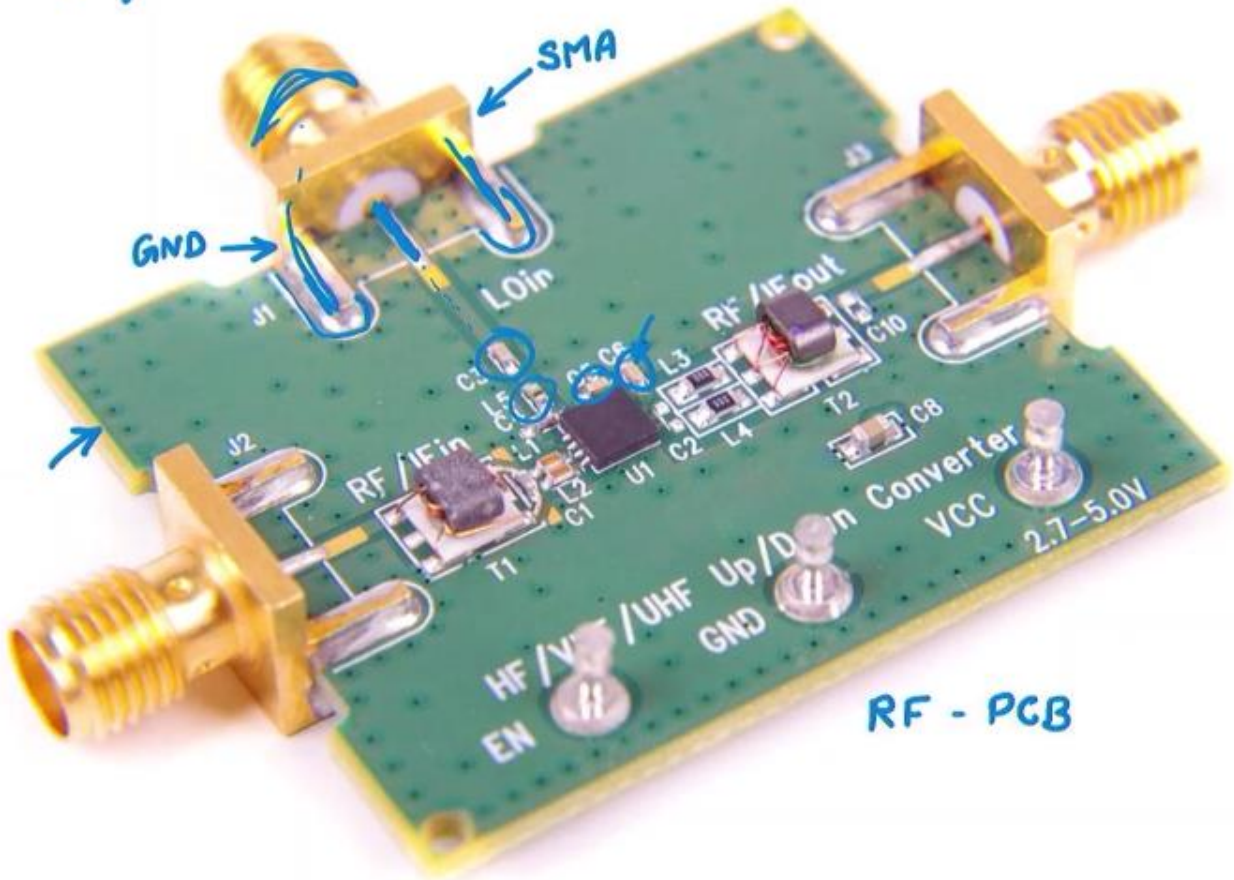
The frequency range for an RF PCB is 300 MHz to 300 GHz.

- Radio frequency (RF) PCBs have become one of the most crucial and commonly used PCBs around the world.
- These are well-known for their capability to handle the unique problems related to transmitting, receiving and processing high-frequency signals.
- RF PCB design plays a significant role in the development of wide range of high frequency applications, like wireless communication systems, radio devices etc.
- Earlier were limited to military and aerospace applications, but now commonly used in consumer electronics.

RF PCB design is more complex than standard PCB design. It requires careful consideration of following points :

- Material Selection. (depends upon freq.)
- Component selection and placement.
- Temperature control.
- Trace design.
- Impedance matching
- Via size
- Loss tangent
- Grounding techniques to ensure signal integrity.

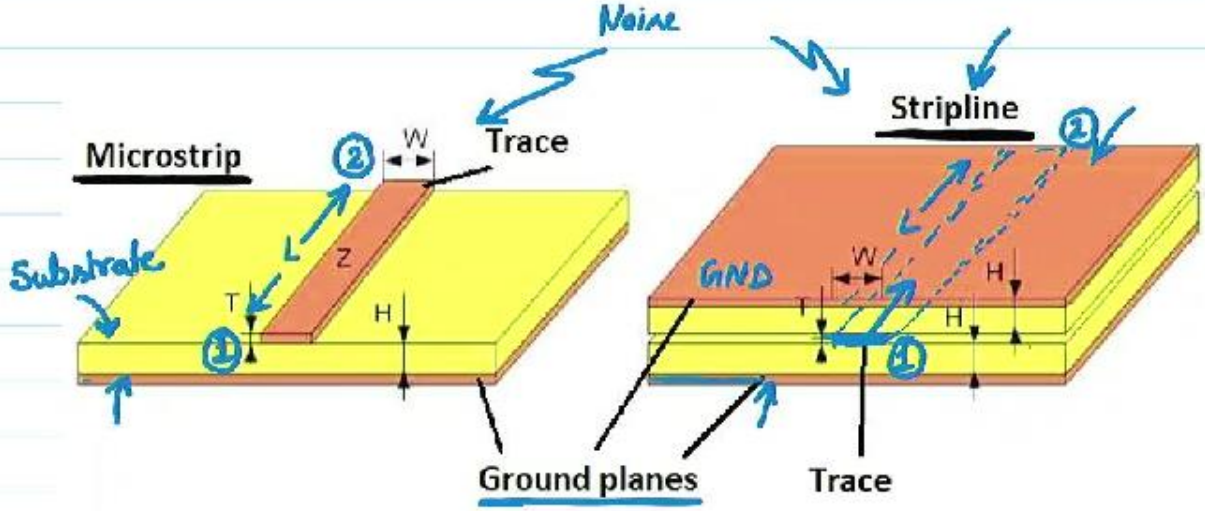
- Every element in the design process impacts the reliability and efficiency of RF System.



RF - PCB

How signal is routed in RF-PCB?

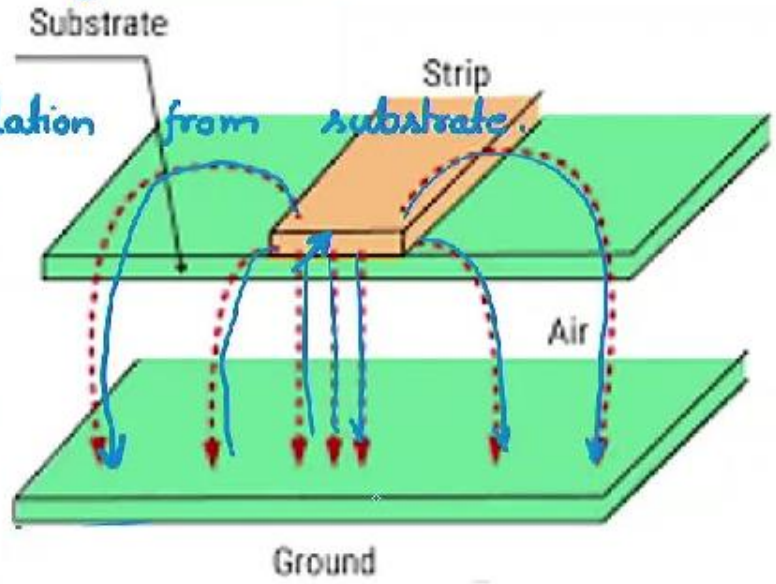
- Stripline and Microstrip are two different transmission line structures used in RF-PCBs.



→ Stripline is fully exposed to di-electric material, dielectric material properties will have greatest influence.

→ Microstrip has lower losses due to isolation from substrate.

→ Stripline has advantage to provide isolation from RF-Noise.



NPTEL

Micro-strip Transmission Line :

- Microstrip transmission lines are popular in RF PCB design due to their simplicity and ease of fabrication.

- They consist of a signal trace on top and a ground plane at the bottom, separated by a di-electric material.

- Trace width, spacing and dielectric constant are critical for achieving the desired characteristic impedance.