1 | Introduction

This project aims to construct an array of antennas for the ESP32 board. The selection of an antenna array is advantageous for various applications, such as directing power to a user through directional antennas. This is achieved by providing the feeder signal to each array element with different phase shifts, optimizing propagation for maximum power transfer in certain important paths or security purposes. Additionally, the array can be utilized for signal diversity, mitigating deep fading in a specific propagation path by compensating with another. However, the ESP32 is typically used as an IoT mobile device where directivity is not essential, the diversity concept can be applied to enhance the signal-to-noise ratio. It's crucial to note that the feeder signal must reach each array element with the same phase shift.

2 | Why PCB antennas?

PCB antennas are highly advantageous, providing essential features such as compactness and lightweight construction. These characteristics make them particularly suitable for applications with space limitations or where minimizing weight is paramount. The integration of the antenna directly onto the PCB is a key benefit, eliminating the necessity for additional external components. This not only simplifies the design process but also contributes to a cost-effective solution.

The feasibility of PCB antennas comes from the prevalent use of gigahertz carrier frequencies in our daily lives. These frequencies enable the utilization of RF transmission lines, allowing for the design of antennas directly printed onto PCBs. Additionally, the construction of matching networks is facilitated through simple strip lines on the PCB. This utilization of gigahertz frequencies and RF transmission lines enhances the practicality of PCB antennas, offering a versatile and efficient solution for various electronic applications.

3 | The Antenna design used

We will use an Inverted-F antenna for each element of the 2 X 2 antenna array, those elements are coupled by an RF power combiner.

3.1 | Inverted - F Antenna

This PCB antenna is used on the CC2511 USB dongle reference design is described in this Section. Even if the antenna presented is for a USB dongle design it can be used in all 2.4 GHz designs, especially where small space is required for the antenna.

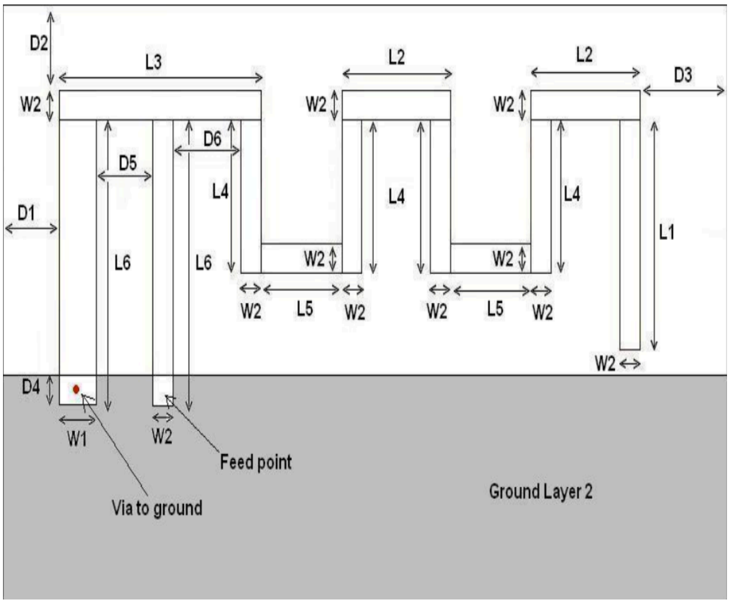
The suggested antenna design requires no more than 15.2 x 5.7 mm of space and ensures a VSWR ratio of less than 2 across the 2.4 GHz ISM band when connected to a 50 ohm source.

The PCB antenna on the CC2511 USB dongle reference design is a meandered Inverted F Antenna (IFA). The IFA was designed to match an impedance of 50 ohm at 2.45 GHz. Thus no additional matching components are necessary. (Antenna impedance is 50 ohms at 2.45GHz - our source at the esp32 pin has 50 ohm impedance)

The inverted-F antenna was first conceived in the 1950s as a bent-wire antenna. However, its most widespread use is as a **planar inverted-F antenna** (**PIFA**) in mobile wireless devices for its space-saving properties.

The inverted-F antenna is suited for esp32 because it has a 50 ohm impedance, omnidirectional that is needed for mobile applications.

3.2 | Inverted F model and dimensions from the reference described above



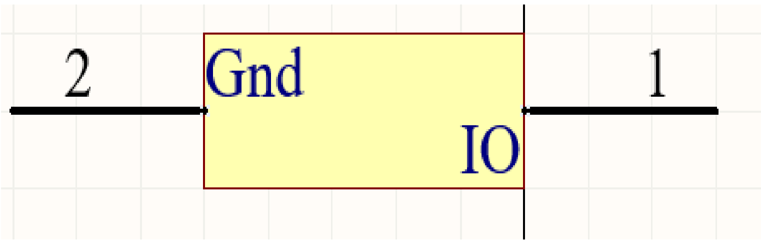
L1	3.94 mm
L2	2.70 mm
L3	5.00 mm
L4	2.64 mm
L5	2.00 mm
L6	4.90 mm
W1	0.90 mm
W2	0.50 mm
D1	0.50 mm
D2	0.30 mm
D3	0.30 mm
D4	0.50 mm
D5	1.40mm
D6	1.70 mm

Notes

- The shorting arm, the most left, connects to the ground as an inductive stub to cancel the antenna capacitive effect of the L-part of the antenna. providing 50 ohm antenna.
- No grounding plane should interfere with the inverted F structure to have omnidirectional desired propagation.

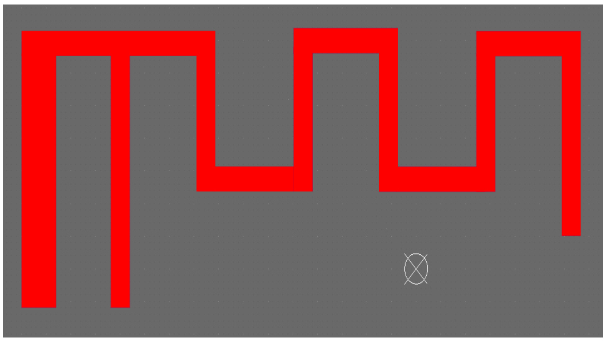
4 | The Altium footprint of the Antenna

We need to make the antenna modular/component as copper fill might introduce problems in clearances or short circuits, especially for PCB - printed Antennas that appear as DC short ckt, we can avoid that by assigning a net type to our component in the design. so we included a schematic library assigned to the PCB library with a footprint that includes the PCB antenna Traces, the antenna needs a gnd and input/output to the esp32, the following shows the process:

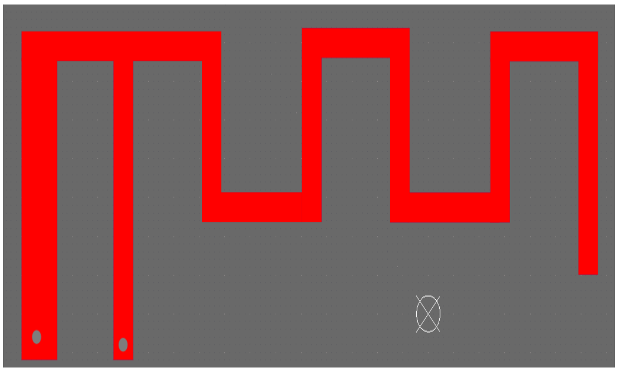


A (Net Tie no BOM) design that indicates no external component is needed to assemble by the manufacturer.

First, we draw the antenna shape with its dimensions in the top layer, after that we want the antenna copper to be exposed to air for the highest gain possible so we apply the Antenna shape to the top solder layer. we also added pads to be able to connect the antenna to the rest of the PCB by assigning the pads to pins 1 and 2 in the schematic.



TOP - LAYER



TOP PASTE

TOP SOLDER

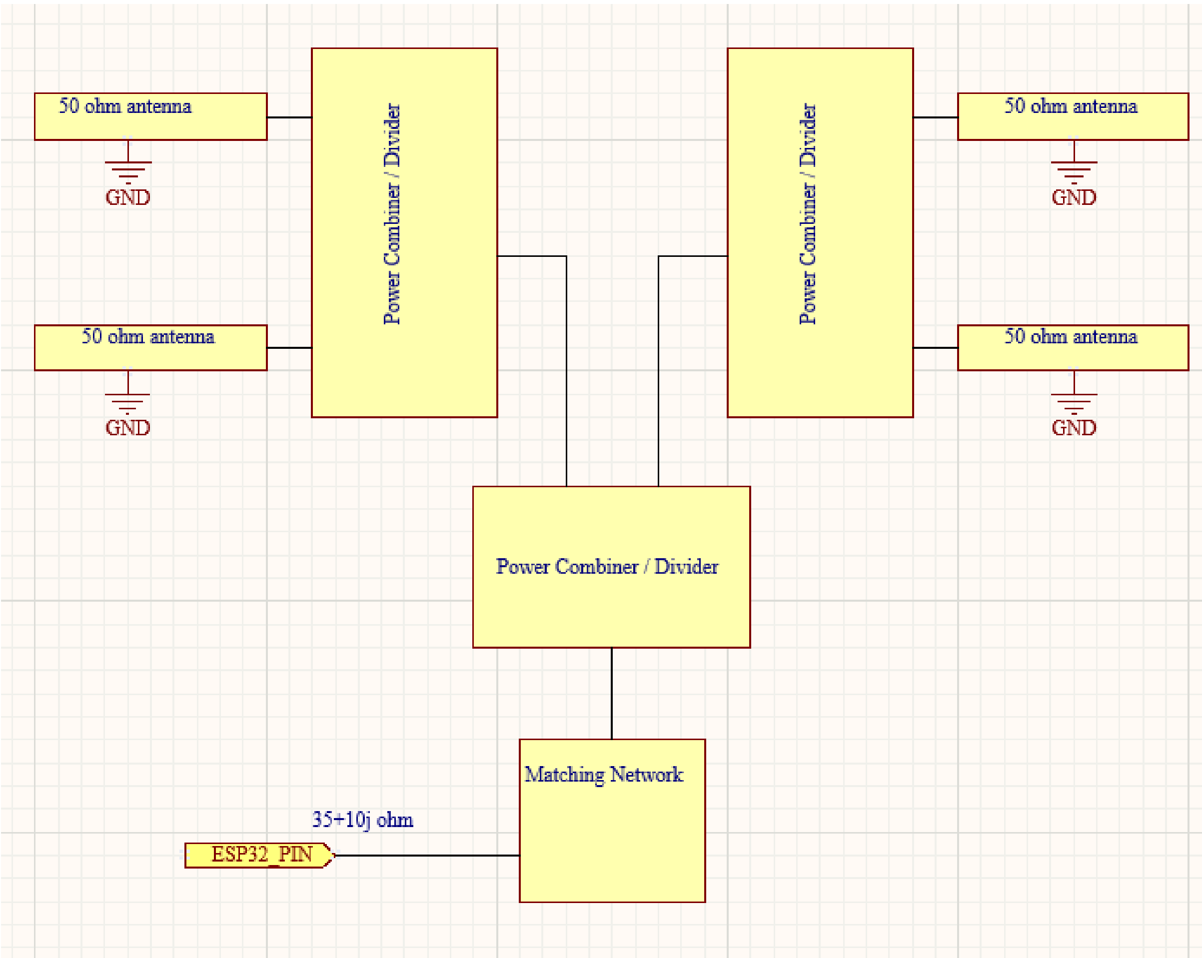
5 | Power Combiner / Divider for the array

An important component in the design because it can combine incoming signals and propagate the input signal from the esp32 RF pin to the four Antennas, Important features in power dividers and combiners are their ability to transfer power and combine power efficiently (By providing proper matching to each port - eliminate reflections - and low power dissipation in the process - lossless -).

Another important feature in power combiners is that we want the power to be split and combined in a split ratio, my design utilizes an equal split power divider. Hence, we need a power combiner/divider for each two-antennas of the four antennas connected. the combiner will ensure an equal split of inserted power at high efficiency.

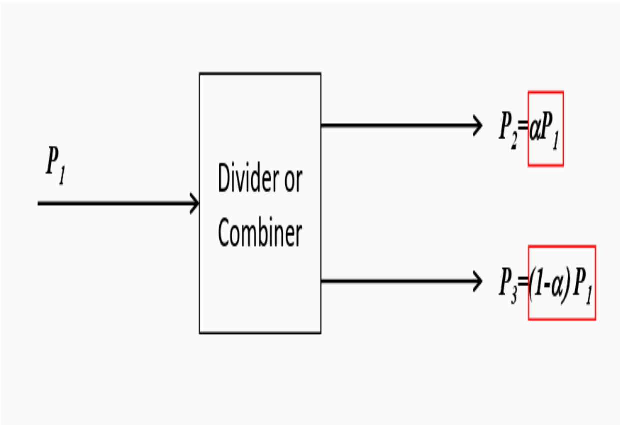
5.1 | System design

We have four Antennas connected as 4X4 Array using power combiners/dividers along with a PI-Matching Network to match the 4 Array network impedance to the ESP32 RF pin impedance of 35+10j ohms as follows:



Design Illustration

6 | Power combiner/divider



A device that will split the power From an incoming pin or combine the power coming from two pins to one pin.
To achieve maximum power split we use power some sort of matching so that at any pin will experience equal output impedance to its own. hence at proper matching, lossless power divider $\alpha = 0.5$.

Many designs implement power division and combining usually trade-offs are made between those designs :

1. **Resistive power divider** (Compromise power efficiency due to lossy resistive components used in series for space compactness -suitable in IC design where wafer space is cost-critical).
2. **Wilkinson power divider** (Compromise space for power efficiency by using lossless transmission lines with parallel resistors).

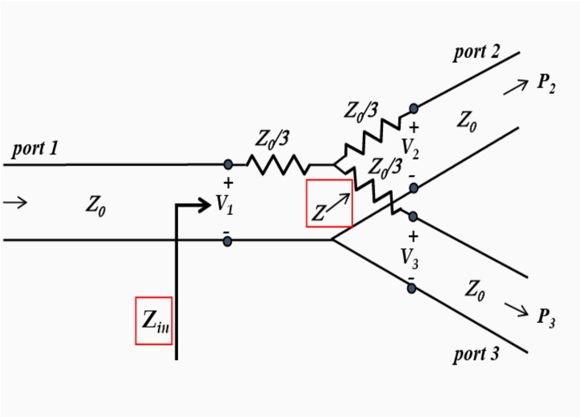
6.1 | Design 1 Resistive power dividers

A resistive power divider can be seen by its port scattering matrix, for example, a 3 X 3 scattering matrix for a 3-port network is seen as follows.

$$[S] = \begin{bmatrix} S_{11} & S_{12} & S_{13} \\ S_{21} & S_{22} & S_{23} \\ S_{31} & S_{32} & S_{33} \end{bmatrix}$$

For maximum power transfer, reflections (i.e. S_{ii}) should equal zero. Hence, we want to achieve S11=S22=S33=0. this can be met at proper matching.

Matching a 3 port resistive power divider



Note that

$$Z_{in} = \frac{Z_0}{3} + \frac{Z}{2}$$

$$Z = \frac{Z_0}{3} + Z_0 = \frac{4Z_0}{3}$$

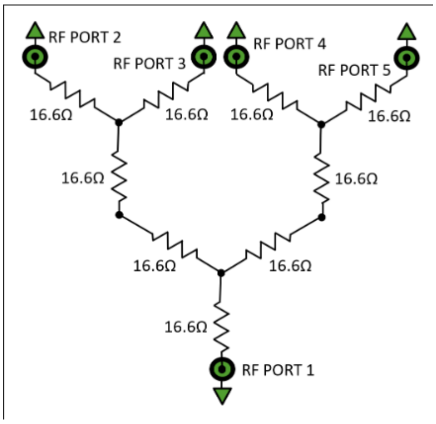
Hence,

$$Z_{in} = \frac{Z_0}{3} + \frac{2Z_0}{3} = Z_0$$

Port 1 is matched and due to the symmetry of ports, all other ports are matched.

Notes

- A design problem of the resistive combiner in the power loss can be solved by amplifiers. but this design is very convenient as it provides good matching and a very compact design. for the 5-port resistive power divider received power is 1/16 of transmitted.
- A design for 5 - port resistive divider:

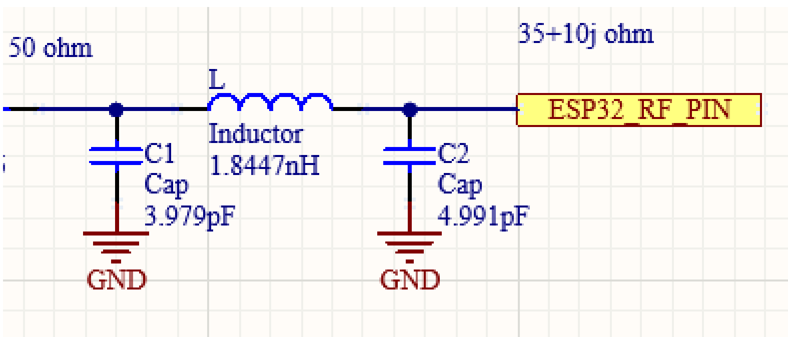


RF-PORT 1 is connected to the esp RF pin. the esp pin has an impedance that needs to be matched with the port RF1 impedance of:

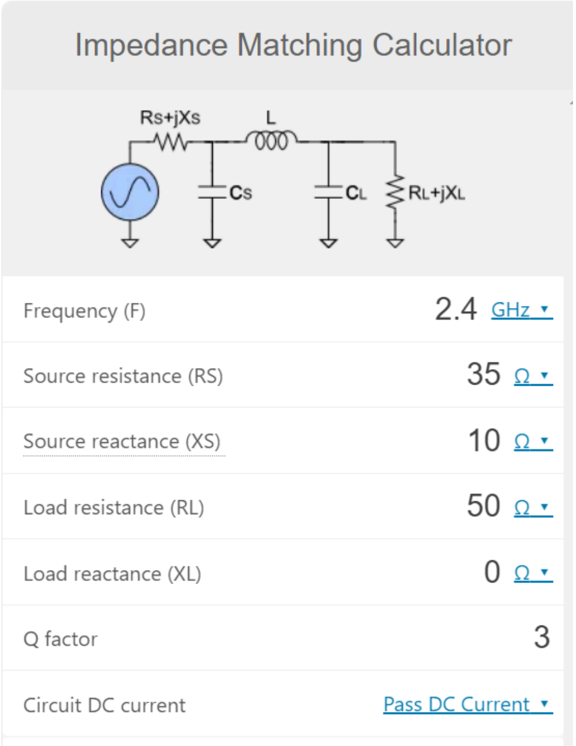
$$Z_{RF1} = 16.6 + \frac{16.6+16.6+\frac{16.6+50}{2}}{2} = 50\ ohm$$

This Matching to the esp pin of 30+10j ohms can be done by a PI-matching network.

PI - Matching Network to the esp-pin



The PI-network values are calculated by the online calculator for simplicity and faster calculation as follows.

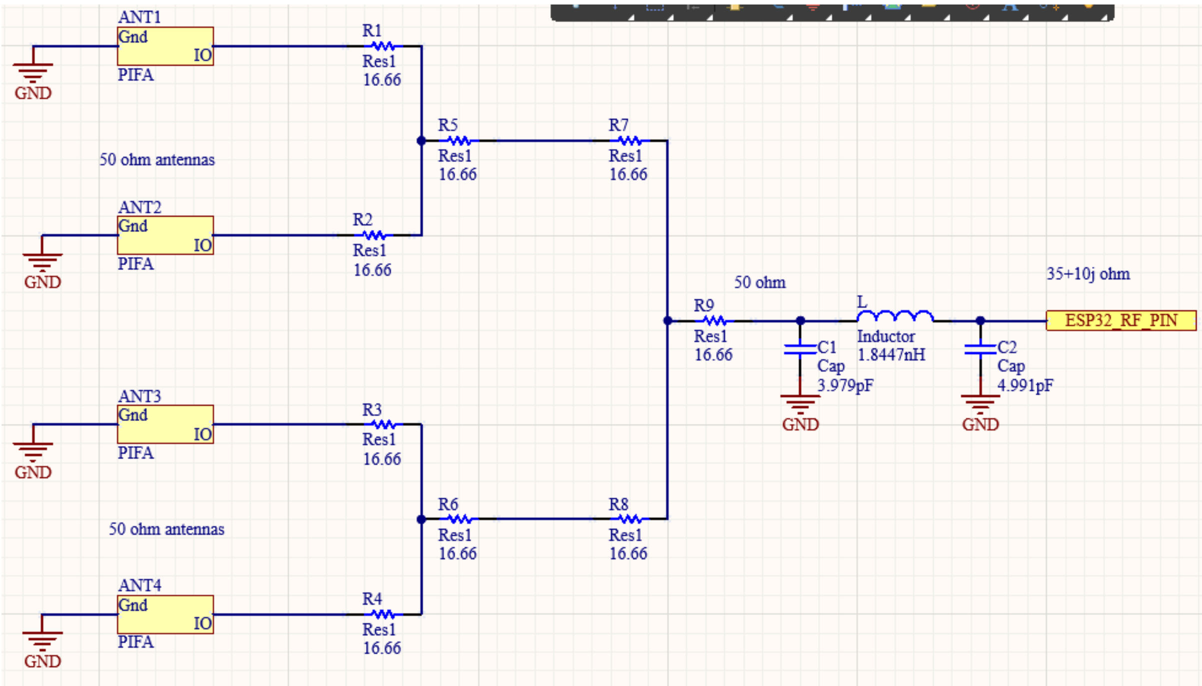


2.4GHz the WiFi carries frequency. esp pin impedance of 30+10j ohms and load (Antenna Array impedance) of 50 ohms. the Q factor controls the operating bandwidth so the lower Q the higher the bandwidth but we will suffer from a lower gain of the network.

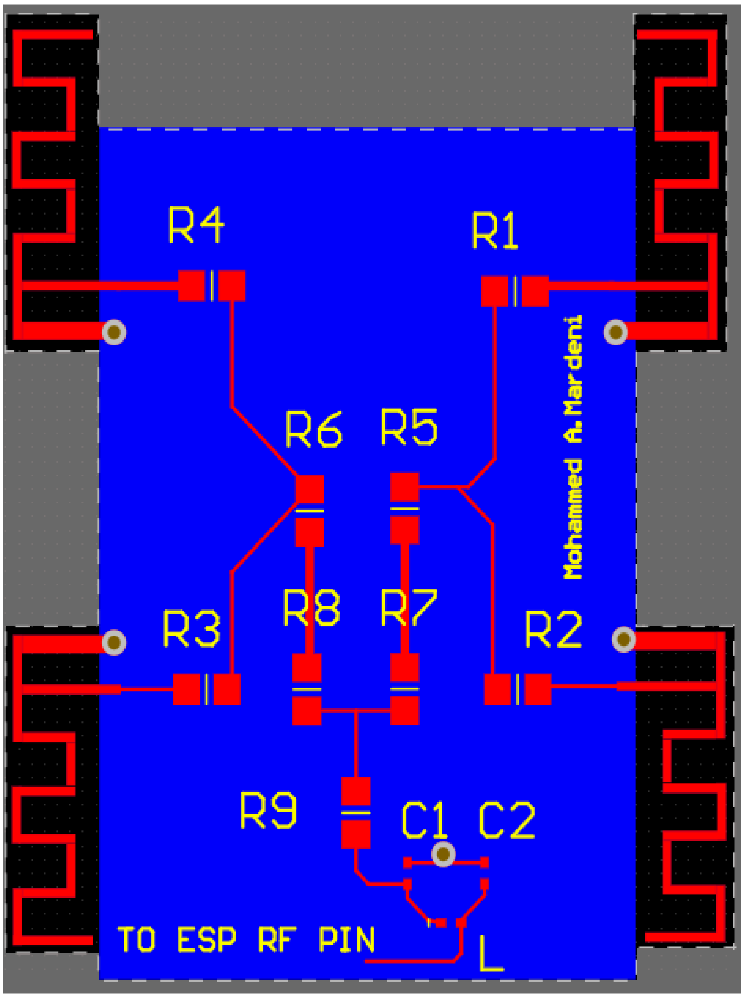
Results of the pi-matching network calculator

Inductance (L)	1.8447	nH
Source capacitance (CS)	4.991	pF
Load capacitance (CL)	3.979	pF

- **Schematic design for a resistive power combiner:**



- **Implementing the design on the PCB as follows:**



Considerations:

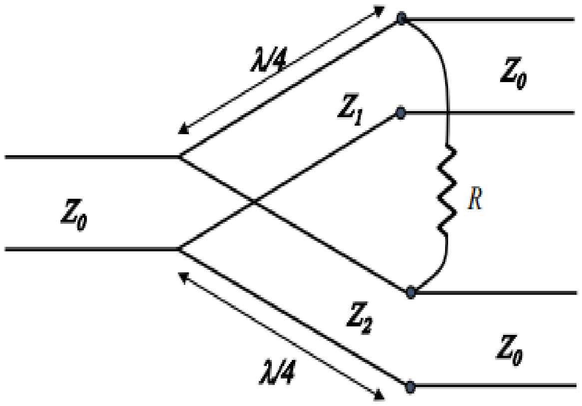
Ensure equal spacing of Antennas and equal route distances so that all route signals have the same phase hence they don't cancel each other.

Introducing a grounding plane to reduce noise and interference, while keeping the antennas out of the grounding plane to have a good Omnidirectional radiation pattern.

The grounding plane connected to the shorting leg of the antennas and the capacitor grounding legs via the plane by using vias assigned to the plane net (gnd).

"A better design for more efficient power utilization and is more suited for PCBs is The Wilkinson power Divider / Combiner."

6.2 | Design 2 Wilkinson power Divider/Combiner



Wilkinson uses lossless quarter wave length transformer transmission lines along with parallel line-line impedance, this will ensure low power loss with excellent matching characteristics, but it takes more space. hence, this divider is usually implemented in PCBs rather than ICs.

Ideally Wilkinson power divider has the following advantages:

- Good matching at all ports
- Low loss
- High isolation between port 2 and port 3

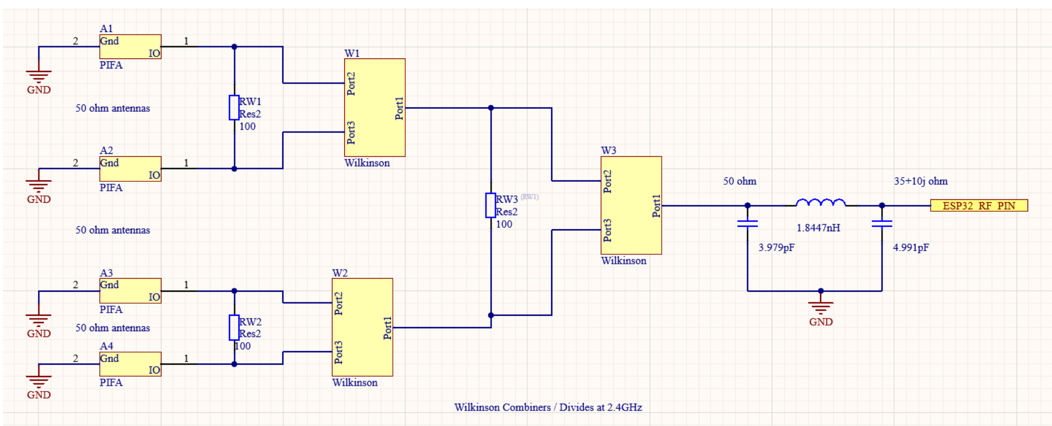
But practically:

- In practice, losses will occur (typ. 1 dB) and the isolation will be limited (typ. -20 dB).
- Practical designs will also be frequency dependent (typ. 20% bandwidth).

Typical Wilkinson divider/combiner modeling approach:

- Use odd-even mode technique, for symmetrical split Z_1=Z_2.
- Resistor, $R = 2Z_0 = 100$.

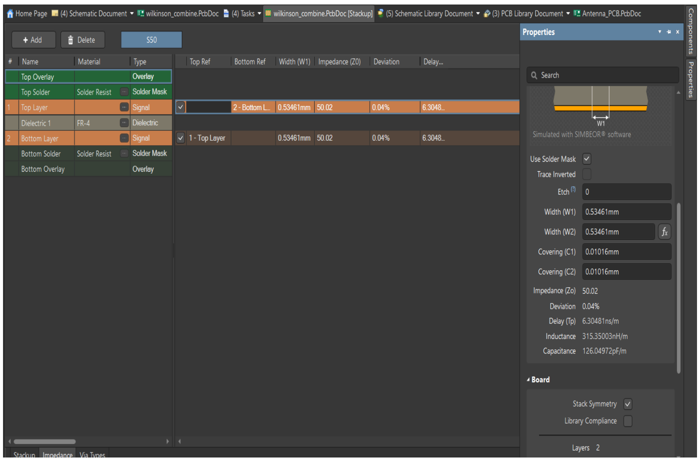
- **Schematic of the system using the Wilkinson:**



- **Implementing Wilkinson 3 port combiner:**

The transmission line:

Velocity of the wave: $\frac{1}{\sqrt{LC}}$ from the layer stack manager we can include an impedance profile, by default it will add 50-ohm characteristic impedance for the layers, we can check for C and L from properties of the layer as follows.



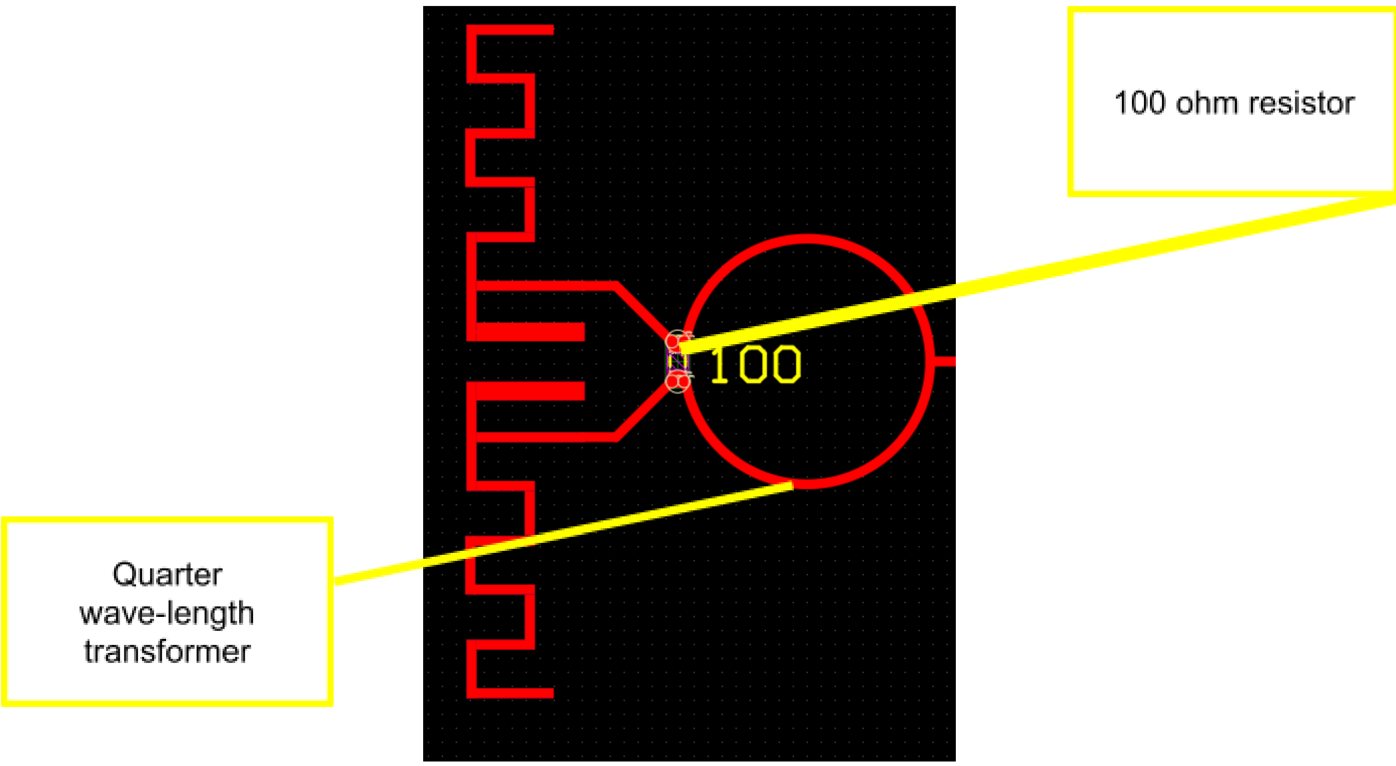
Note that the reference should be the bottom layer.

The L and C: 315nH/m, 126pF/m.

Now we can calculate quarter wave length of:

$$\frac{\lambda}{4} = \frac{V}{f\ x\ 4} = \frac{1}{4\ x\ 2.4GHz\ X\ \sqrt{LC}} = 16.5219mm$$

- **The design of the 3 port Wilkinson:**



- **Implementing 5 port Wilkinson combiner:**