RF CAD LAB REPORT

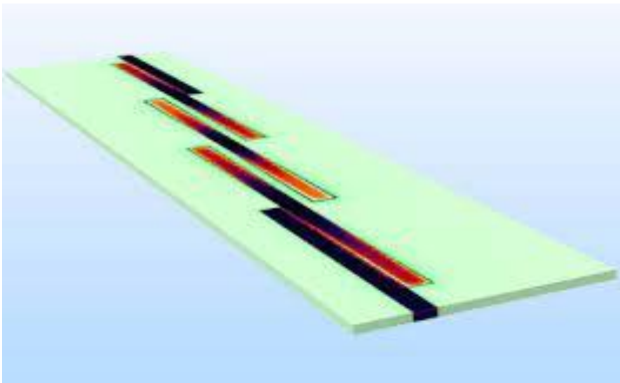
Submitted by

Rudraprasad Debnath
MTech RF and Microwave Engineering
Roll EE22M113
2nd Year 1st Semester

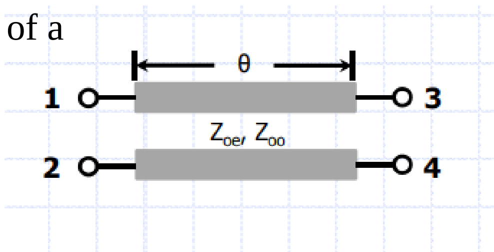
Contents

Experiment 1: Coupled Line Bandpass Filter (Design and Fabrication)

Introduction: Coupled line Bandpass filters are one of the most popular and useful planar microwave filters those are used in many fields. This type of filters are designed with a series of parallel microstrip lines those appear as a pair and are parallel to each other. Each pair corresponds to a combination of lumped elements (inductors and capacitors), those can be used to realize a filter response. Each section has an **even impedance** and an **odd impedance.**



Theory: From the figure shown in the diagram of a parallelly-coupled line, first the Z-matrix is to be calculated. Z_{oe} = even mode characteristic impedance, Z_{oo} = odd mode characteristic impedance



$$\mathbf{Z_{11}=Z_{22}=Z_{33}=Z_{44} = \frac{-j}{2}(Z_{oe}+Z_{oo})\cot\theta}$$

$$\mathbf{Z_{12}=Z_{21}=Z_{34}=Z_{43} = \frac{-j}{2}(Z_{oe}-Z_{oo})\cot\theta}$$

$$\mathbf{Z_{13}=Z_{31}=Z_{24}=Z_{42} = \frac{-j}{2}(Z_{oe}+Z_{oo})\csc\theta}$$

$$Z_{14}=Z_{41}=Z_{23}=Z_{32} = \frac{-j}{2}(Z_{oe}-Z_{oo})\csc\theta$$

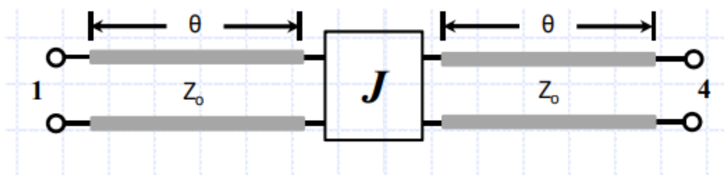
Thus, the impedance matrix

$$\begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} = \begin{bmatrix} \frac{-j}{2}(Z_{oe}+Z_{oo})\cot\theta & \frac{-j}{2}(Z_{oe}-Z_{oo})\cot\theta \\ \frac{-j}{2}(Z_{oe}+Z_{oo})\csc\theta & \frac{-j}{2}(Z_{oe}-Z_{oo})\csc\theta \end{bmatrix}$$

And from the impedance matrix, we can calculate the ABCD matrix

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} \frac{Z_{11}}{Z_{41}} & \frac{|Z|}{Z_{41}} \\ \frac{1}{Z_{41}} & \frac{Z_{44}}{Z_{41}} \end{bmatrix}$$

$$\begin{bmatrix} A & B \\ C & D \end{bmatrix} = \begin{bmatrix} \frac{Z_{oe}+Z_{oo}}{Z_{oe}-Z_{oo}}\cos\theta & j\frac{(Z_{oe}-Z_{oo})^2 csc^2\theta - (Z_{oe}+Z_{oo})^2 cot^2\theta}{2(Z_{oe}-Z_{oo})\csc\theta} \\ 2\frac{j}{Z_{oc}-Z_{sc}}\sin\theta & \frac{Z_{oe}+Z_{oo}}{Z_{oe}-Z_{oo}}\cos\theta \end{bmatrix}$$



For the lines shown in the above figure,

$$[ABCD] = [ABCD_{left}][J][ABCD_{right}]$$

$$= \begin{bmatrix} \cos\theta & jZ0sin\,\theta \\ j\frac{1}{Z0}sin\,\theta & cos\,\theta \end{bmatrix}\begin{bmatrix} 0 & -j\frac{1}{J} \\ -jJ & 0 \end{bmatrix}\begin{bmatrix} cos\,\theta & jZ0sin\,\theta \\ j\frac{1}{Z0}sin\,\theta & cos\,\theta \end{bmatrix}$$

$$= \begin{bmatrix} \left(jZ_0 + \frac{1}{jZ_0}\right) \sin\theta\cos\theta & j(JZ_0{}^2 \sin^2\theta - \frac{\cos^2\theta}{J}) \\ j(\frac{1}{jZ_0{}^2} \sin^2\theta - J\cos^2\theta) & \left(jZ_0 + \frac{1}{jZ_0}\right) \sin\theta\cos\theta \end{bmatrix}$$

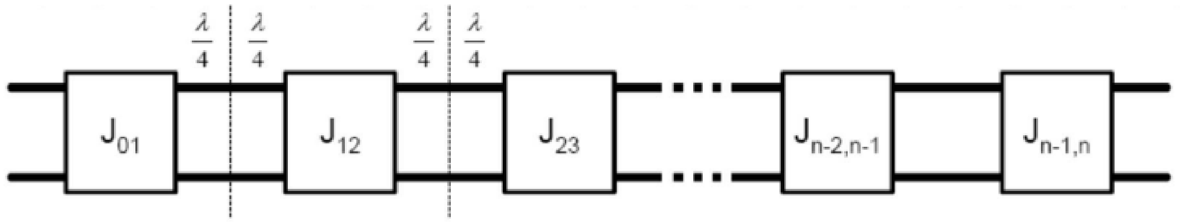
Comparing the A and C terms, when $\theta \approx \frac{\pi}{2}$

$$\frac{Z_{oe} + Z_{oo}}{Z_{oe} - Z_{oo}} = \left(jZ_0 + \frac{1}{jZ_0}\right)$$

And

$$\frac{Z_{oc} - Z_{sc}}{2} = jZ_0{}^2$$

$$\boxed{\begin{aligned} &\Rightarrow Z_{oe} = Z_0(1 + JZ_0 + J^2 Z_0{}^2) \\ &\Rightarrow Z_{oo} = Z_0(1 - JZ_0 + J^2 Z_0{}^2) \end{aligned}}$$



For the design shown in the above figure,

$$\frac{J_{n,n+1}}{Y_0} = \sqrt{\frac{\pi FBW}{g_n g_{n+1}}}$$

Where, FBW = Specified fractional bandwidth of the filter

g_n = normalized parameter values obtained from the Chebyshev response chart

For a Hairpin BPF, there are some separate parameters those need to be defined.

K_n is the coupling coefficient between the resonators that can be defined as,

$$\boldsymbol{K_n = \frac{FBW}{\sqrt{g_k g_{k+1}}}}$$

Design and Simulation: In the design procedure, a 4th order filter with 1.69 GHz center frequency was planned to be designed that will have a Chebyshev response. The Chebyshev response table with 0.5 dB ripple was studied to extract the normalized parameter values of the filter. The values were obtained as,

$$g0 = 1.000$$
$$g1 = 1.5963$$
$$g2 = 1.0967$$
$$g3 = 1.5963$$
$$g4 = 1.000$$

After that, the even and odd mode impedances of each coupled line section was calculated using the formulae shown in the previous sections.

n	g_n	Z_0J_n	$Z_{0e}(\Omega)$	$Z_{0o}(\Omega)$
1	1.5963	0.3137	70.61	39.23
2	1.0967	0.1187	56.64	44.77
3	1.5963	0.1187	56.64	44.77
4	1.0000	0.3137	70.61	39.23

Now, from a commercial calculator, the length, width and the gap of each coupled line pair was calculated,

Stage	W(mm)	L(mm)	S(mm)
1	2.35588	25	0.459
2	2.89136	24	1.734
3	2.89136	24	1.734
4	2.35588	25	0.459

Now, the filter is designed in a full wave EM simulator (Ansys HFSS) on a substrate of FR4 Epoxy (ϵ_r = 4.4, h = 1.6mm), and the simulated results are shown below. The simulated results show a good resemblance with the desired results. The filter is also fabricated in the lab and measured in a VNA machine. The results also match well with the simulated outputs. Two 50Ω microstrip lines are later connected at the both ends of the designed filter.

Results:

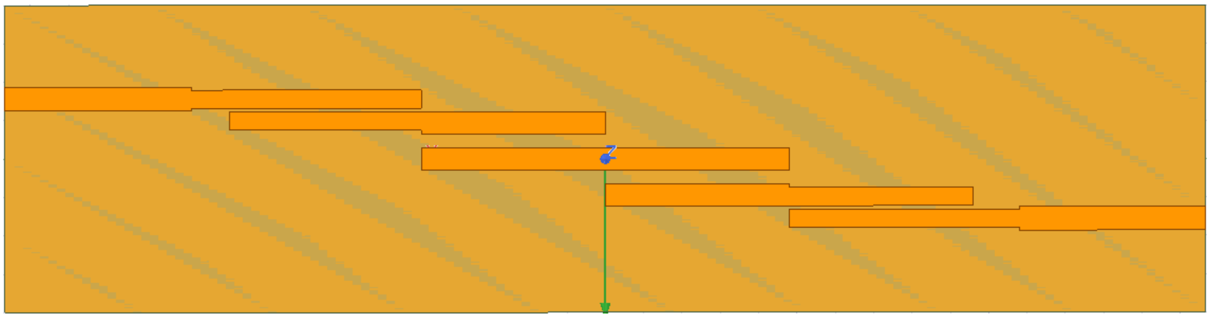


Figure 1: Designed filter in HFSS

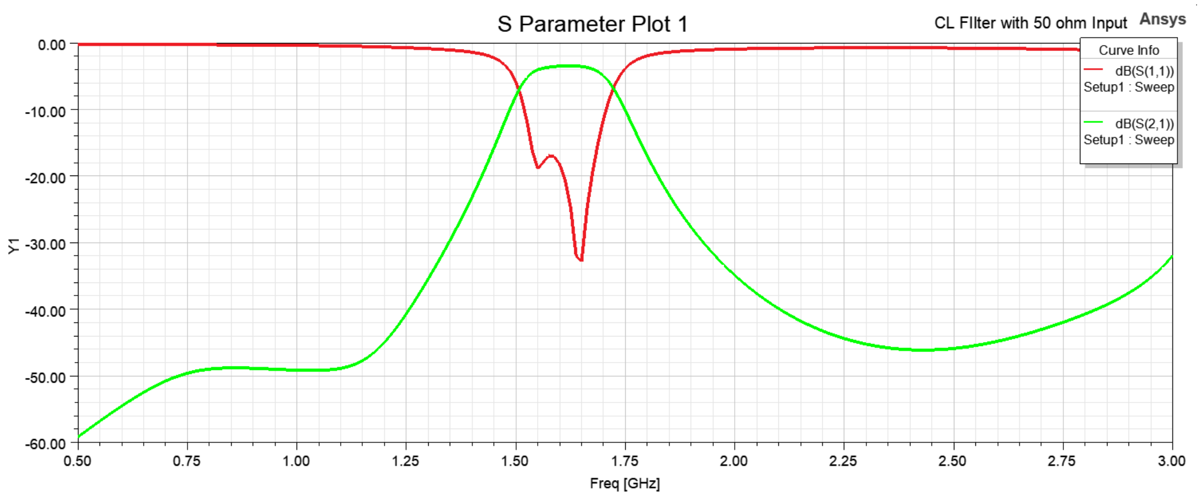
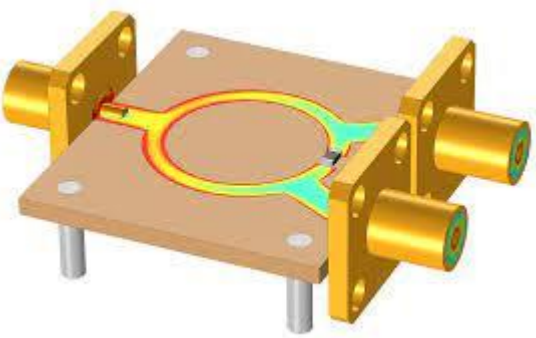


Figure 2:S11 and S21 parameter plot of the filter

Fabrication:

The fabricated filter is shown below,

Experiment 2: Equal Split Wilkinson Power Divider



Introduction: Power dividers are one of the most frequently used microwave component used in RF circuitry. As in RF design, power is more emphasized than voltage, power dividers play an important role. Power dividers can be of many types, it can be a simple resistive divider or a T-junction waveguide divider. But according to Microwave theory, no component can be matched, lossless and reciprocal at the same time. So, in Wilkinson power divider, a separate technique is used. A lossy element such that a resistor is added in the circuit, but due to symmetry, no current flows through it. Thus, the component can be made lossless, matched and reciprocal at the same time. The value of the resistor is chosen depending on the characteristic impedance of the microstrip lines used.

Theory: The figure aside shows the circuit diagram of the Wilkinson power divider. Even and odd mode analysis gives the S-parameter matrix of the device as,

$$S = \begin{matrix} 0 & \frac{-j}{\sqrt{2}} & \frac{-j}{\sqrt{2}} \\ \frac{-j}{\sqrt{2}} & 0 & 0 \\ \frac{-j}{\sqrt{2}} & 0 & 0 \end{matrix}$$

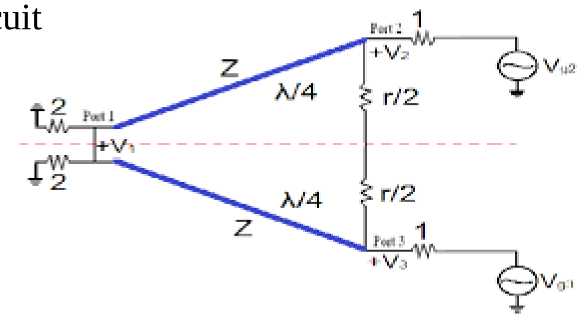


Figure 3: Wilkinson Power Divider

From the matrix it can be concluded that there must be equal split of power from the port 1 to each of the ports 2 and 3, and also there will be a phase shift of 90 degrees between them. Also, there must be a high isolation between the ports 2 and 3 that is achieved by the resistive element put between the ends of two transmission lines of length λ/4. The matrix is obtained by Even and Odd mode analysis of the given circuit.

Design, Simulation and Results: A Wilkinson power divider is designed in Keysight ADS using the practical microstrip lines (MLIN) and the layout is also simulated in EM Co Simulation. The results are shown in the following figures.

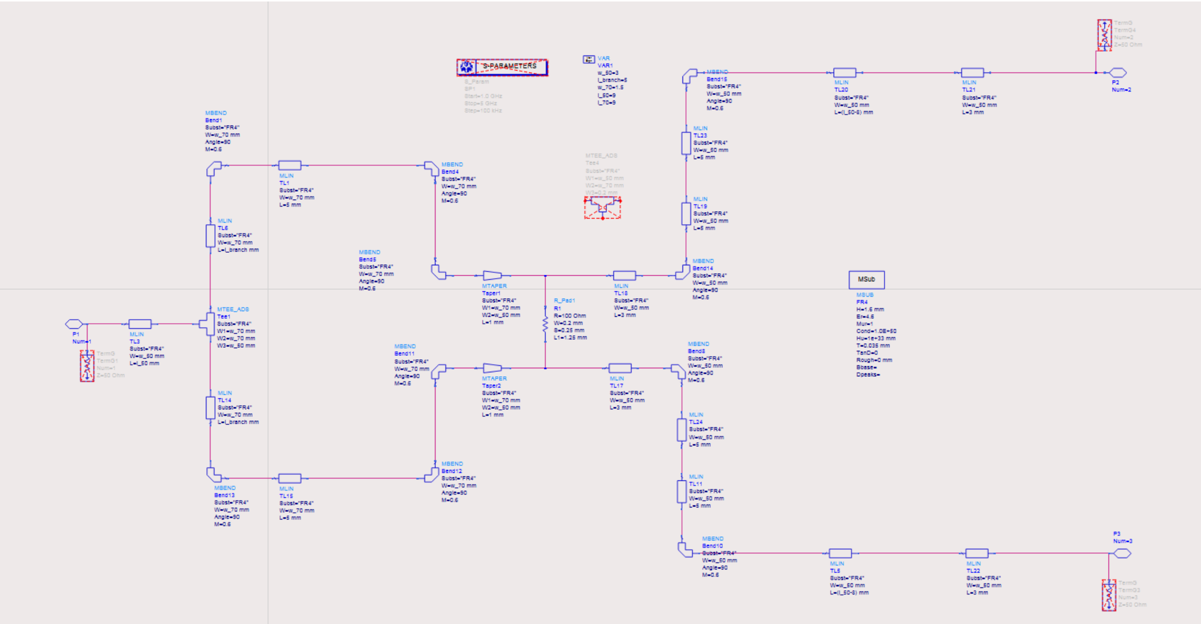


Figure 4: Wilkinson Power Divider Schematic

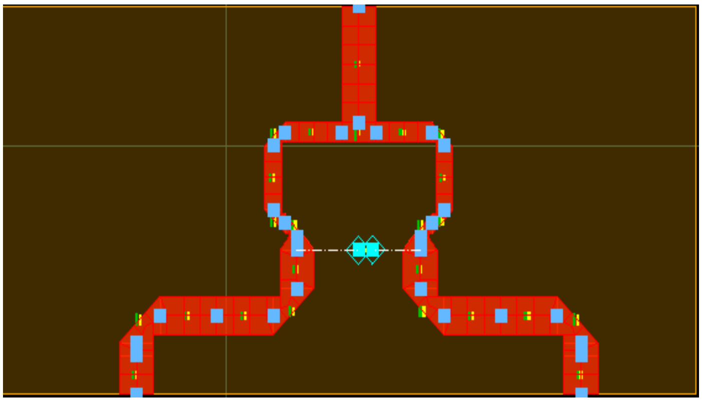
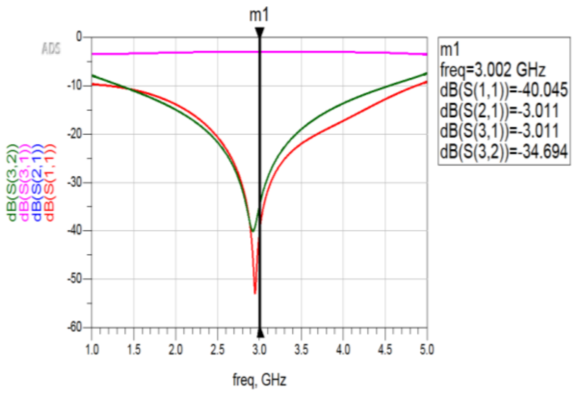
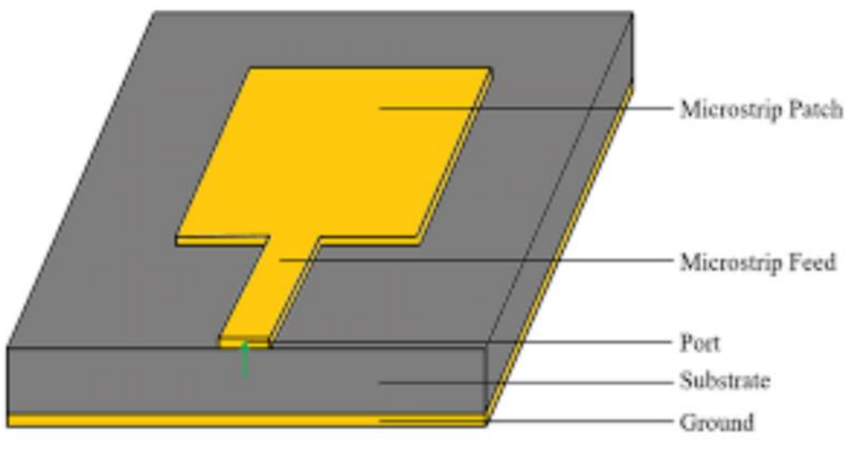


Figure 5: Layout



Experiment 3: Design and analysis of a microstrip patch antenna a) Linearly polarized 2) Circularly polarized

Introduction: Microstrip patch antennas are one of the fastest growing topics in the field of RF and Microwave Engineering. This is a low-profile antenna that can be mounted on the top of the substrate easily and with low cost. They can also be used efficiently dual and triple frequencies, and lower radiation power. As this type of antenna contains only a patch mounted on the substrate, they are also called as “Patch” antennas. In figure 1, the structure of a rectangular microstrip patch antenna is shown. It is clear that it contains three layers – The ground plane (bottom), substrate (middle) and the top patch (top). There is also a feeding line that is necessary for excite the antenna with proper feeding. In this figure, the antenna is fed by a microstrip line. But it can also be fed by coaxial feeds or an Inset feed. To make the antenna work efficiently, the feed must be matched to the edge impedance of the patch.



For circular polarization, the tip of the electric field of the propagated wave follows a circular locus. To make a patch antenna circularly polarized, we need to excite two orthogonal modes simultaneously, and for that purpose, two feeds are used at two positions 90 degrees apart and two excitations must also have 90 degrees phase difference.

Design and Simulation: A rectangular patch antenna is designed in Ansys HFSS software with two orthogonal feeds and 90 degree phase difference. FR4 epoxy is used as the substrate and simulated with an input power of 1 Watt. The arrangement is shown in the following diagram.

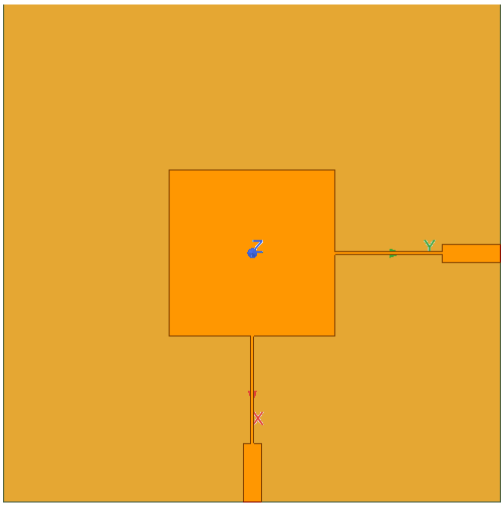


Figure 7: Rectangular Patch Antenna

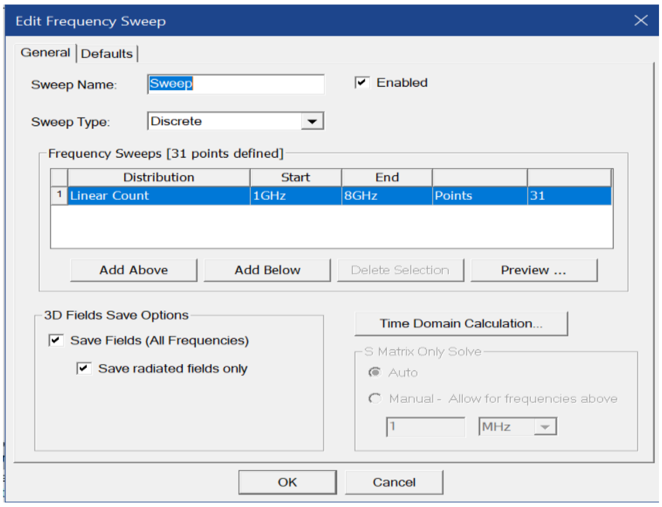


Figure 8: Sweep Setup

Results: The simulation results are presented in the below diagram.

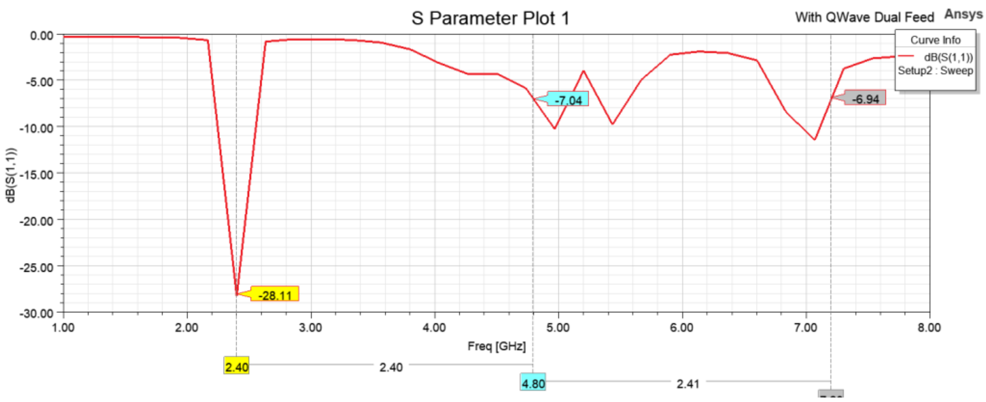
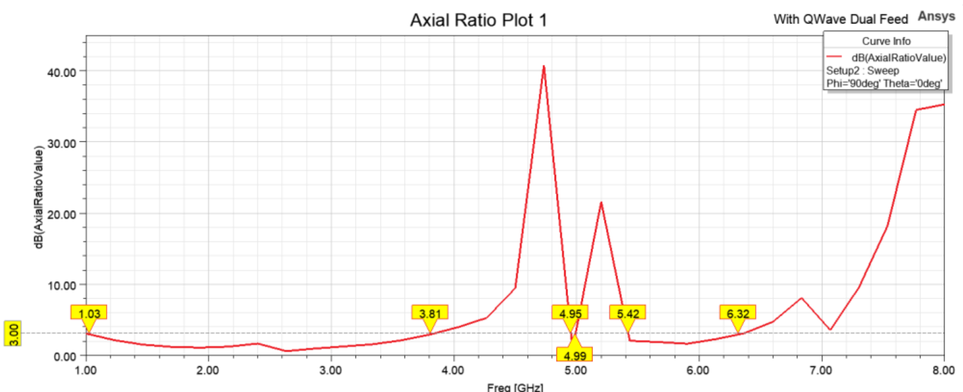


Figure 9: S-Parameter Plot



From the axial ratio plot it is clear that the axial ratio is less than 3 dB from 1 GHz to 4 GHz range that signifies that the antenna is circularly polarized.

Another result for the linearly polarized patch antenna is shown below—

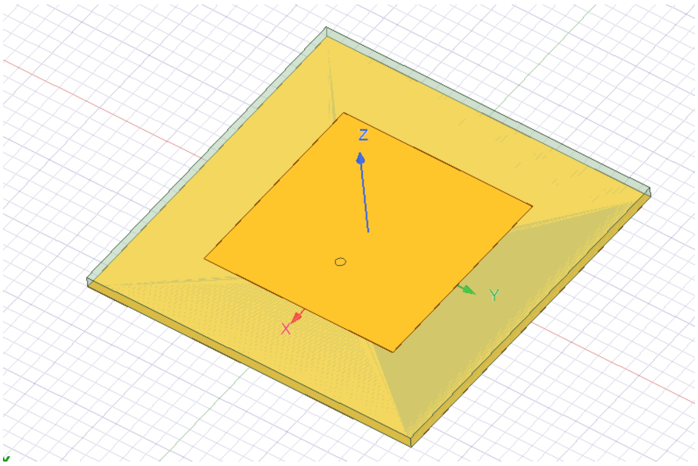


Figure 11: Linearly polarized Patch Antenna

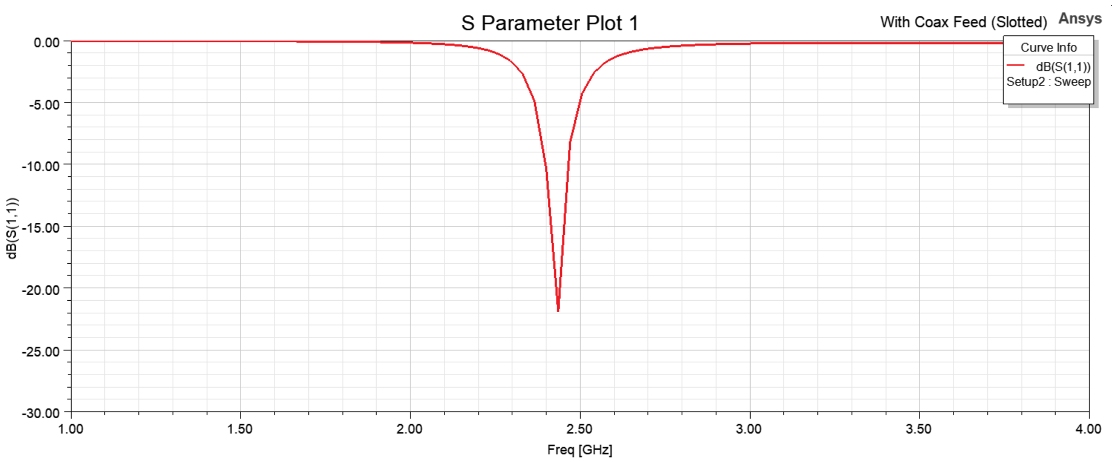


Figure 12: S-Parameter plot

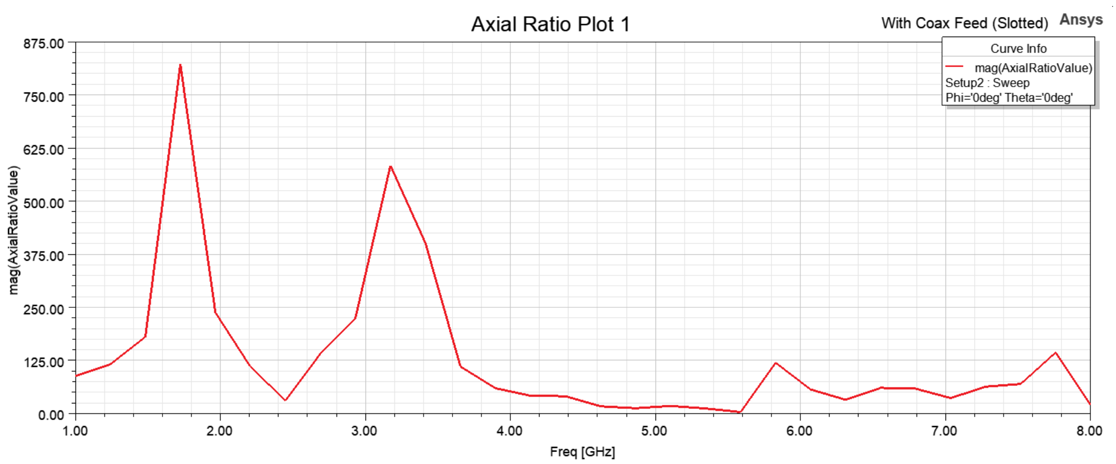


Figure 13:Axial Ratio Plot

Experiment 4: Substrate Integrated Waveguide (SIW)

Introduction: A **substrate-integrated waveguide** (SIW) is a synthetic rectangular electromagnetic waveguide formed in a dielectric substrate by densely arraying metallized posts or via holes that connect the upper and lower metal plates of the substrate. The waveguide can be easily fabricated with low-cost mass-production using through-hole techniques, where the post walls consists of via fences. SIW is known to have similar guided wave and mode characteristics to conventional rectangular waveguide with equivalent guide wavelength.

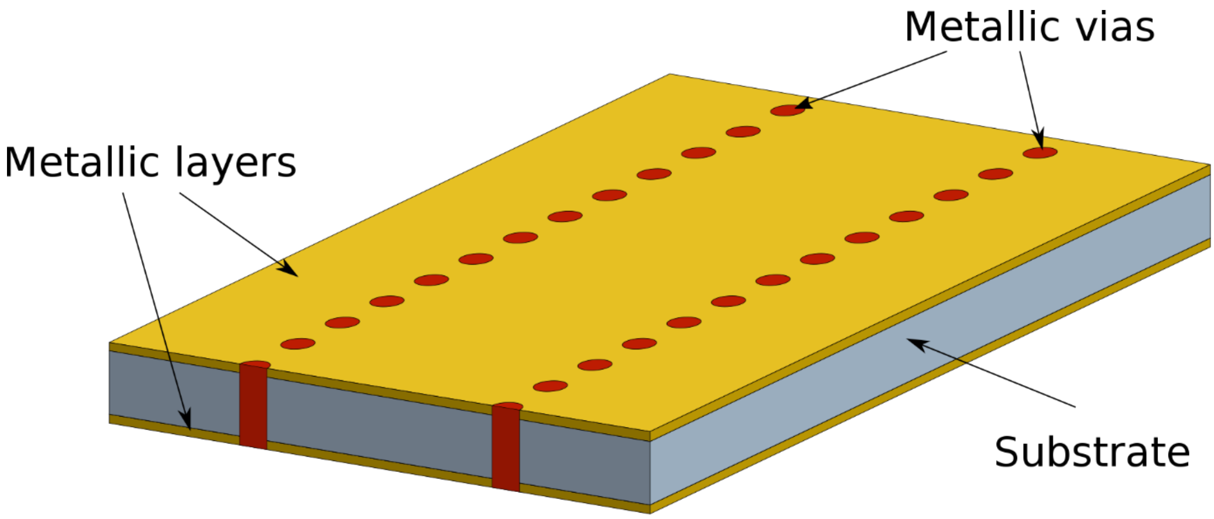


Figure 14: SIW

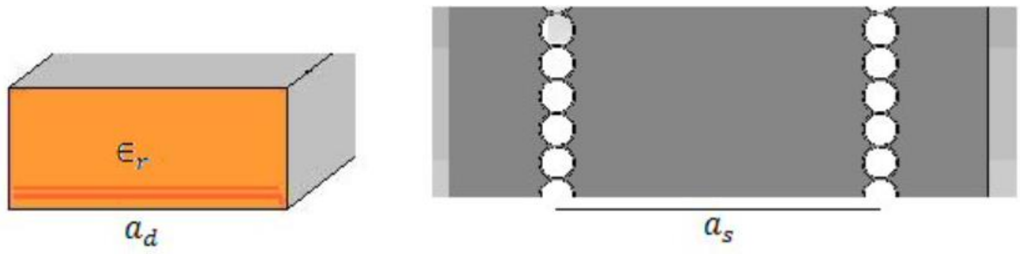
SIWs are used mostly in applications where waveguides are needed but the high-frequency appliances are small in size. Mostly in 5G communication it is used widely. The vias work as the sidewalls of the waveguide. One attraction to SIW is that the amount of metal that carries the signal is far greater than it would be in microstrip or stripline. Therefore, conductor loss α_C is lower. But one disadvantage is that applying dielectric to the SIW increases the dielectric loss. Leakage losses may also be significant.

Design Equations: TE_{10} mode is the dominant mode of the SIW. So, the height of the SIW does not affect the cutoff frequency. We know, the cutoff frequency is given as,

$$f_{c10} = \frac{c}{2a}$$

Where, c = speed of light

a = width of the SIW



For a dielectric filled waveguide, the width becomes,

$$a_d = \frac{a}{\sqrt{\varepsilon_r}}$$

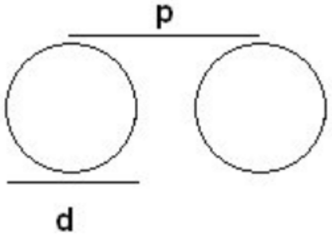
ε_r = Relative permittivity of the dielectric

Having known the value od a_d, the value of a_s, that is the gap between via rows can be determined as below-

$$a_s = a_d - \frac{d^2}{0.95p}$$

Where, p = Pitch, the distance between the vias

d = Diameter of the vias



Design in HFSS: Using the aforementioned equations, a SIW is designed in Ansys HFSS and simulated. Results are shown below. The results seems similar to that of a rectangular waveguide.

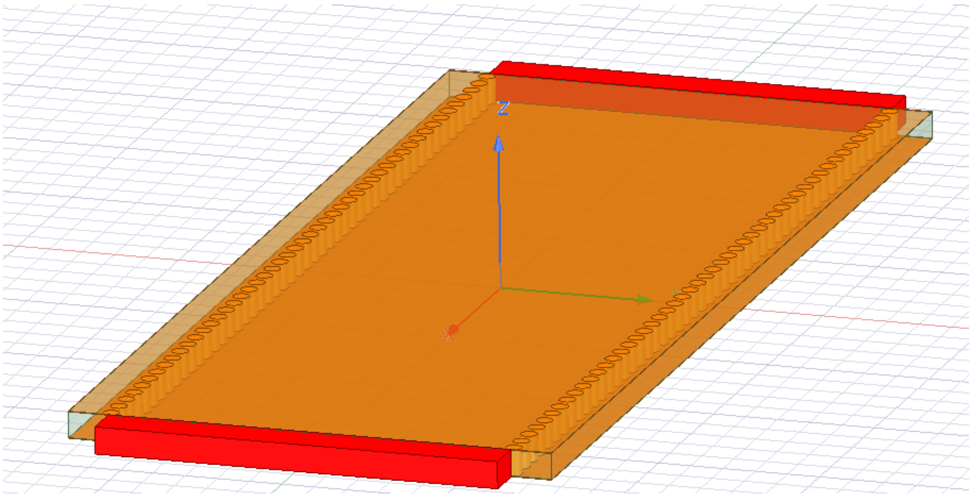


Figure 15: SIW Designed in HFSS

Results: The s-parameter plot of the simulated SIW is shown here,

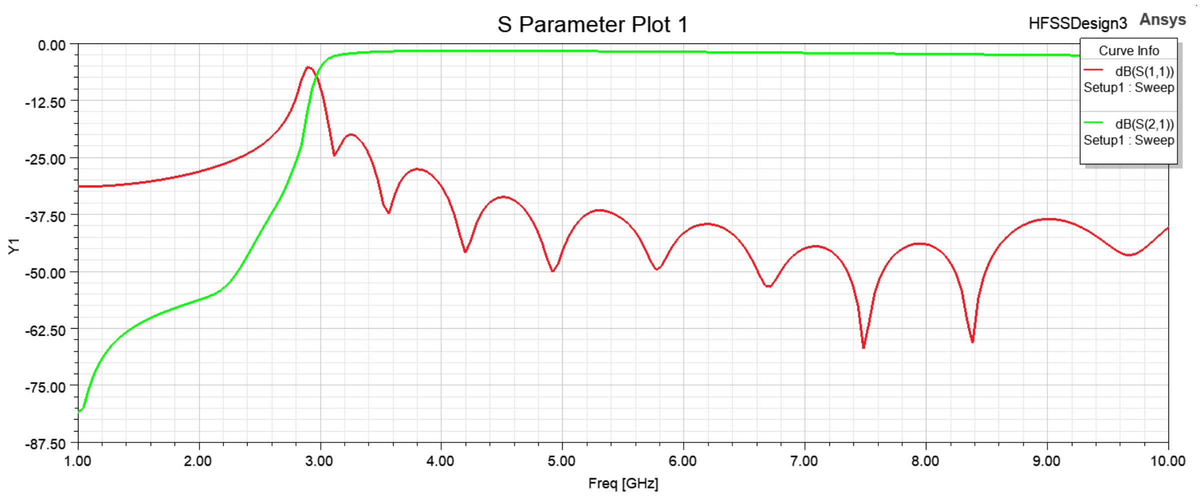


Figure 16:S-parameter plot

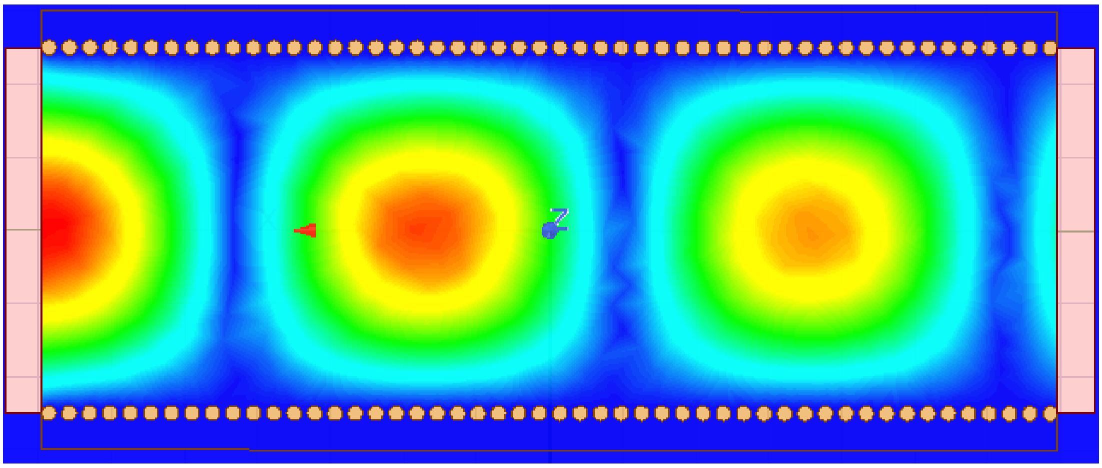


Figure 17: Electric Field Going Through SIW

Experiment 5: Design of Binomial Transformer

Introduction and Theory: Impedance matching is a crucial part of RF design as it is essential to eliminate reflections and allow maximum power to be delivered. For that purpose, several impedance transformers are used and Binomial transformers are one of them. For a given number of sections, the response of this transformer is as flat as possible near the design frequency. This type of response, which is also known as maximally flat, is determined for an N-section transformer by setting the first N − 1 derivatives of |Γ(θ)| to zero at the center frequency, f0. Such a response can be obtained with a reflection coefficient of the following form:

$$\mathit{\Gamma}(\theta) = A(1 + e^{-2j\theta})^N$$

Where, N is the number of sections, A is a coefficient. The value of **A** can be found as below after doing some analysis –

$$A = 2^{-N}\frac{Z_L - Z_0}{Z_L + Z_0}$$

The fractional bandwidth is given as,

$$\frac{\Delta f}{f_0} = 2 - \frac{4}{\pi}\cos^{-1}[\frac{1}{2}(\frac{\mathit{\Gamma}_m}{A})^{1/N}]$$

For n-th section, the reflection coefficient is given as,

$$\Gamma_n = \frac{\mathrm{Z}_{n+1} - \mathrm{Z}_n}{\mathrm{Z}_{n+1} + \mathrm{Z}_n}$$

As Γ’s are assumed to be small, the above expression can be written as,

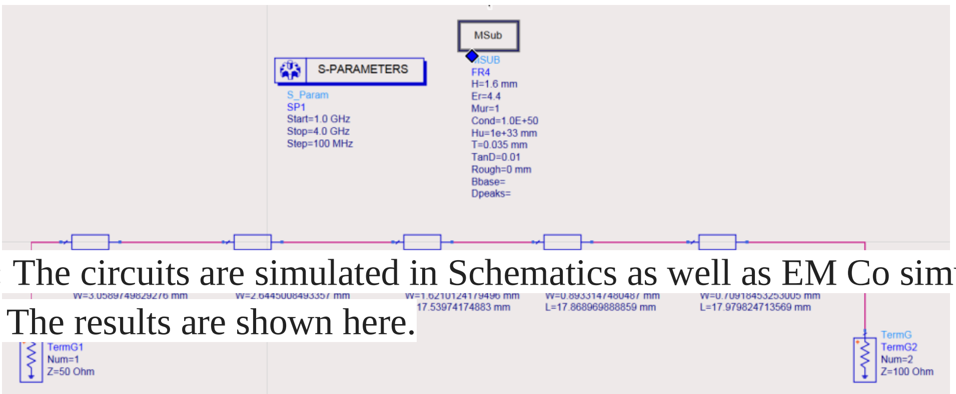
$$\Gamma_n = \frac{1}{2} ln\frac{\mathrm{Z}_{n+1}}{\mathrm{Z}_n}$$

Thus, the impedance for each section can be separately calculated from the above equation.

Design and Simulation: A three-section binomial transformer is intended to design at to match a 50Ω load to a 100Ω load at 2.4 GHz centre frequency. The Γ_m value is given as 0.05, that is the maximum allowable reflection coefficient. The following calculations are made to get the impedances of the respective three sections,

$$
\begin{aligned}
n = 0: \quad \ln Z_1 &= \ln Z_0 + 2^{-N} C_0^3 \ln \frac{Z_L}{Z_0} \\
&= \ln 100 + 2^{-3}(1) \ln \frac{50}{100} = 4.518, \\
Z_1 &= 91.7\ \Omega; \\
n = 1: \quad \ln Z_2 &= \ln Z_1 + 2^{-N} C_1^3 \ln \frac{Z_L}{Z_0} \\
&= \ln 91.7 + 2^{-3}(3) \ln \frac{50}{100} = 4.26, \\
Z_2 &= 70.7\ \Omega; \\
n = 2: \quad \ln Z_3 &= \ln Z_2 + 2^{-N} C_2^3 \ln \frac{Z_L}{Z_0} \\
&= \ln 70.7 + 2^{-3}(3) \ln \frac{50}{100} = 4.00, \\
Z_3 &= 54.5\ \Omega.
\end{aligned}
$$

The length and width of these lines are synthesized with a Microstrip line calculator. Then they are implemented in Keysight ADS.



Results: The circuits are simulated in Schematics as well as EM Co simulation in ADS. The results are shown here.

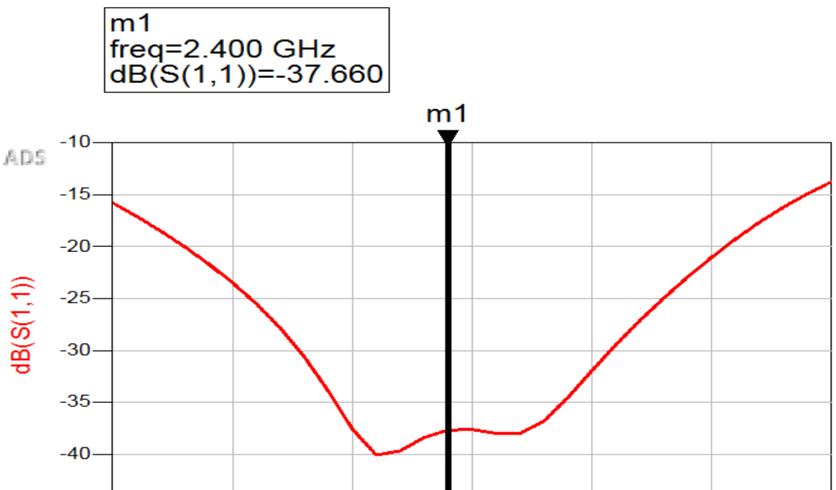


Figure 19: Results from Schematic

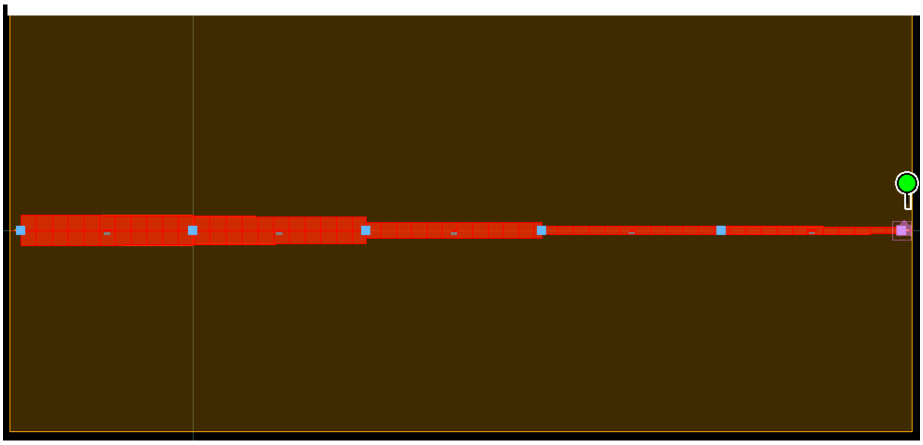
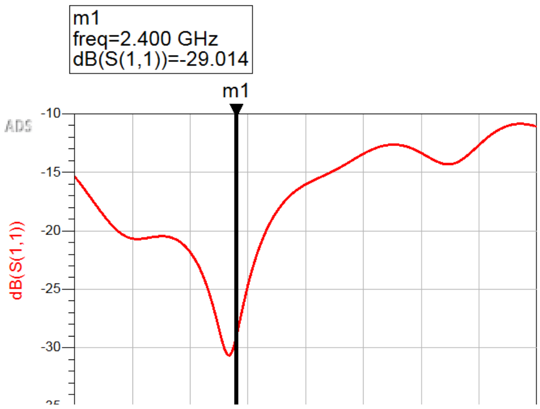


Figure 10:Layout for EM Simulation

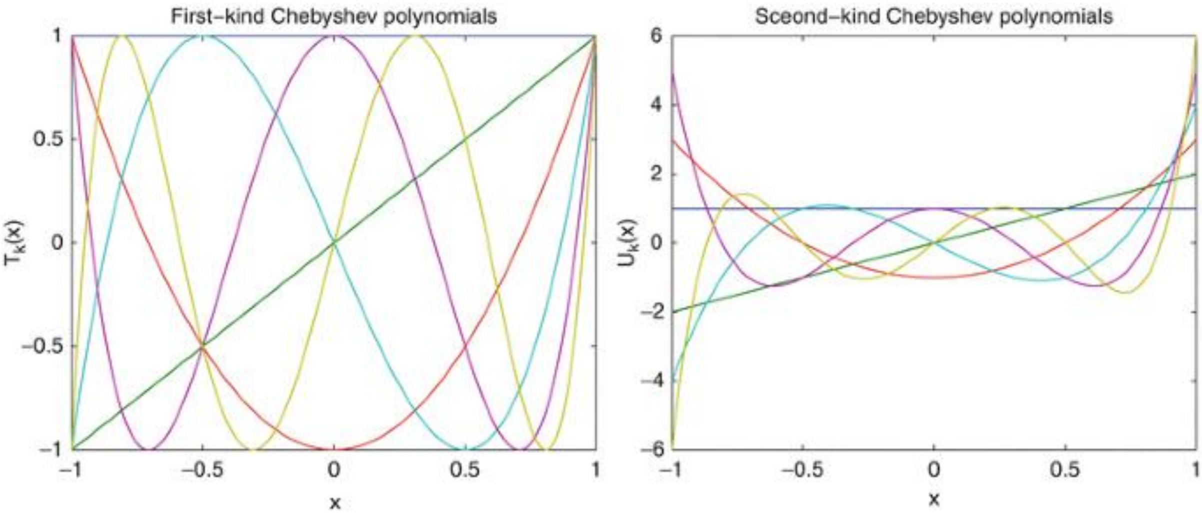


Experiment 6: Design of a Chebyshev Transformer

Introduction and Theory: In contrast with the binomial transformer, the multisection Chebyshev matching transformer optimizes bandwidth at the expense of passband ripple. Compromising on the flatness of the passband response leads to a bandwidth that is substantially better than that of the binomial transformer for a given number of sections. This type of transformers use the Chebyshev polynomials to obtain the matching. Chebyshev polynomials can be explained in the following way—

$$\boldsymbol{T_n(x) = 2xT_{n-1}x(x) - T_{n-2}(x)}$$

The graph can be shown as below



It can be seen that for the input range -1 to 1, the value of the function stays in between -1 and +1 only. So we have to map the desired bandwidth in between this range of the polynomial. Now, let $x = cos\theta$, for |x| < 1. Then,

$$T_n(\cos\theta) = \cos n\theta$$

And the Chebyshev polynomial can be expressed as,

$$T_n(x) = \cos(n\cos^{-1}x) \qquad for \quad |x| < 1$$
$$T_n(x) = \cosh(n\cosh^{-1}x) \qquad for \quad |x| > 1$$

Design Equations: The design equations required for the Chebyshev transformer is shown below—

$$\begin{aligned}\Gamma(\theta) &= 2e^{-jN\theta}[\Gamma_0\cos N\theta + \Gamma_1\cos(N-2)\theta + \cdots + \Gamma_n\cos(N-2n)\theta + \cdots] \\ &= Ae^{-jN\theta}T_N(\sec\theta_m\cos\theta),\end{aligned}$$

Thus,

$$\Gamma(0) = \frac{Z_L - Z_0}{Z_L + Z_0} = AT_N(\sec\theta_m)$$

And,

$$A = \frac{Z_L - Z_0}{Z_L + Z_0} \frac{1}{T_N(\sec\theta_m)}$$

$$T_N(\sec\theta_m) = \frac{1}{\Gamma_m} \left| \frac{Z_L - Z_0}{Z_L + Z_0} \right|$$

And,

$$\sec\theta_m = \cosh\left[\frac{1}{N}\cosh^{-1}\left(\frac{1}{\Gamma_m}\left|\frac{Z_L - Z_0}{Z_L + Z_0}\right|\right)\right]$$
$$\simeq \cosh\left[\frac{1}{N}\cosh^{-1}\left(\left|\frac{\ln Z_L/Z_0}{2\Gamma_m}\right|\right)\right].$$

After θ is known the value of fractional bandwidth can be calculated,

$$\frac{\Delta f}{f_0} = 2 - \frac{4\theta_m}{\pi}$$

Also Γ_n can be approximated as,

$$\Gamma_n = \frac{1}{2} ln \frac{Z_{n+1}}{Z_n}$$

Simulation and Result: In this report, a Chebyshev transformer is intended to design to match a 50Ω impedance to a 100Ω impedance with $\Gamma_m = 0.05$.

From the equation of Chebyshev transformer, we get, for a 3rd order (N=3) transformer,

$$\Gamma(\theta) = 2e^{-3j\theta}(\Gamma_0 cos3\theta + \Gamma_1 cos\theta) = Ae^{-3j\theta}T_3(sec\theta_m cos\theta)$$

A = Γ_m = 0.05 (given)

Then, θ_m can be calculated as, $\theta_m = \sec^{-1} 1.408 = 44.7\ degrees$

Then,

$$2(\Gamma_0 cos3\theta + \Gamma_1 cos\theta) = A(sec\theta_m)^3(cos3\theta + 3cos\theta) - 3Asec\theta_m cos\theta$$

Equating the coefficients of the similar terms,

$$2\Gamma_0 = A(\sec\theta_m)^3$$
$$2\Gamma_1 = 3A((sec\theta_m)^3 - sec\theta_m)$$

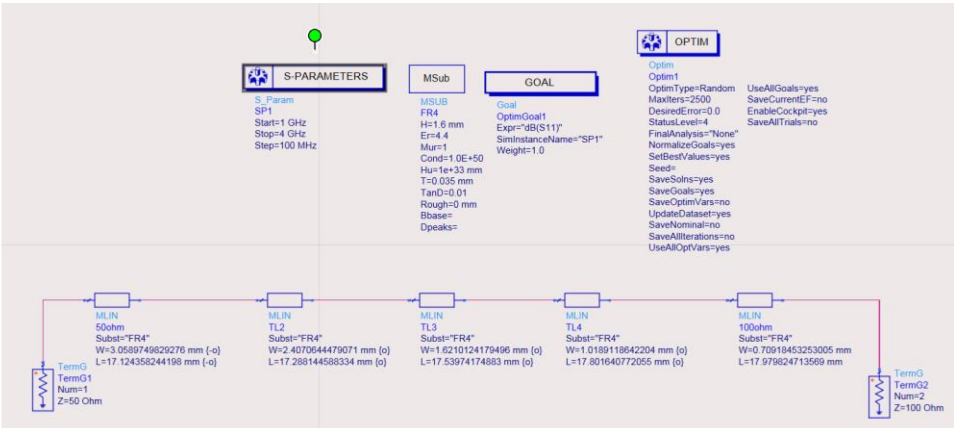
We get,

$$\Gamma_0 = 0.0698$$
$$\Gamma_1 = 0.1037$$

Then the characteristic impedance for the three sections,

$$\begin{aligned}
n = 0: \quad \ln Z_1 &= \ln Z_0 + 2\Gamma_0 \\
&= \ln 50 + 2(0.0698) = 4.051 \\
Z_1 &= 57.5\ \Omega; \\
n = 1: \quad \ln Z_2 &= \ln Z_1 + 2\Gamma_1 \\
&= \ln 57.5 + 2(0.1037) = 4.259 \\
Z_2 &= 70.7\ \Omega; \\
n = 2: \quad \ln Z_3 &= \ln Z_2 + 2\Gamma_2 \\
&= \ln 70.7 + 2(0.1037) = 4.466 \\
Z_3 &= 87.0\ \Omega.
\end{aligned}$$

From a microstrip line calculator, the length and width of the sections are calculated and designed in ADS.



Results: The results obtained after schematic as well as EM simulations are presented—

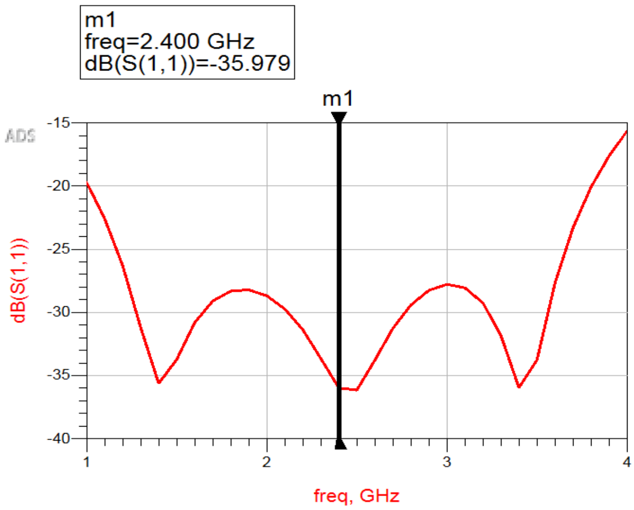


Figure 12: S-parameter curve for schematic

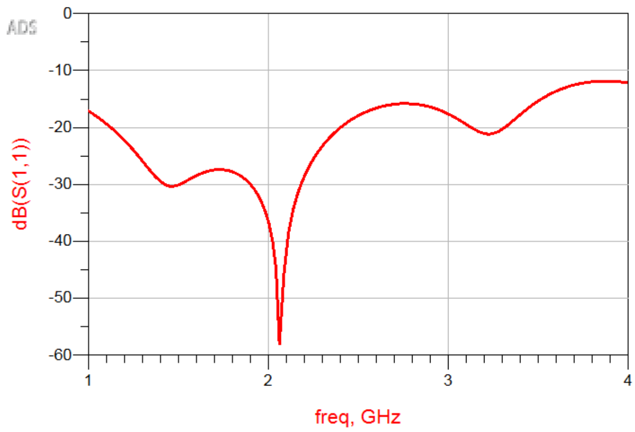


Figure 13:EM Simulation Curve for the Transformer

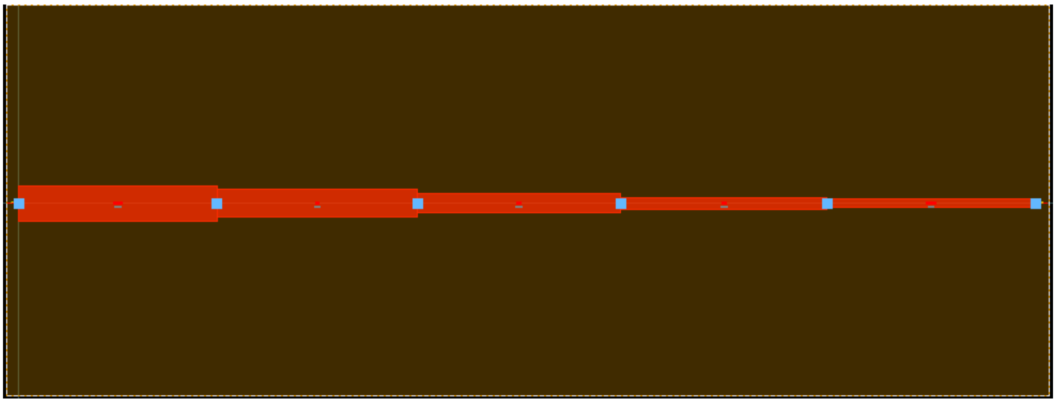
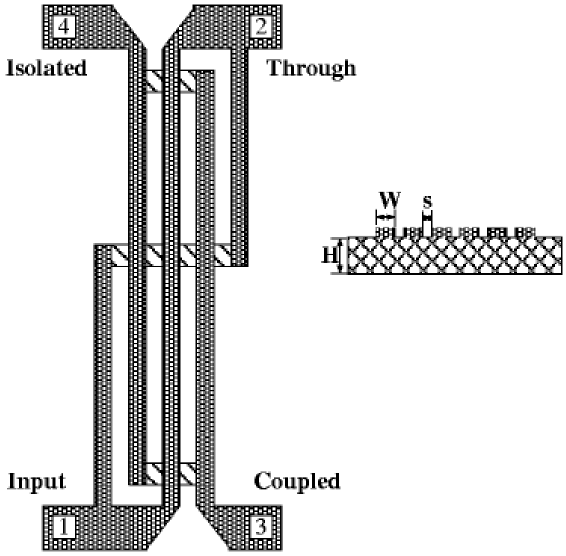


Figure 14:Layout

Experiment 7: Design of a Lange Coupler

Introduction and Theory: Generally, the coupling in a coupled line coupler is too loose to achieve coupling factors of 3 or 6 dB. One way to increase the coupling between edge-coupled lines is to use several lines parallel to each other, so that the fringing fields at both edges of a line contribute to the coupling. One of the most practical applications is the lange coupler. In this type of couplers, four parallel transmission lines are used to get strong coupling.

The figure shows the structure of a lange Coupler. It can be seen that there are interconnects between the alternating lines. The main disadvantage of the Lange coupler is probably practical, as the lines are very narrow and close together, and the required bonding wires across the lines increases complexity.



Let, the voltage coupling coefficient be **C**
For the lange coupler,

So, the even and odd mode impedances for each coupled pair can be calculated as,

$$Z_{0e} = \frac{4C - 3 + \sqrt{9 - 8C^2}}{2C\sqrt{(1-C)/(1+C)}} Z_0$$

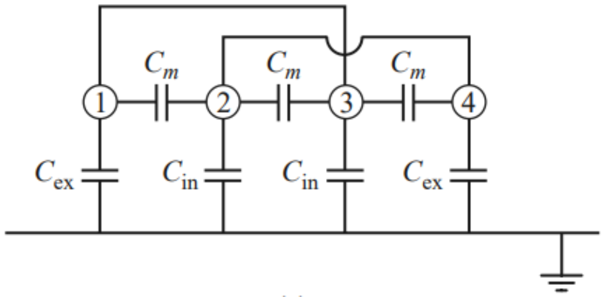
$$Z_{0o} = \frac{4C + 3 - \sqrt{9 - 8C^2}}{2C\sqrt{(1+C)/(1-C)}} Z_0$$

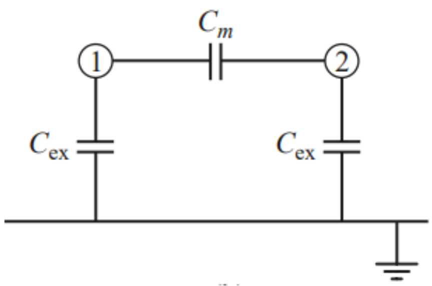
From this, the even and odd impedance of the equivalent two-conductor lines of the lange coupler can be measured as,

$$Z_{e4} = \frac{Z_{0o} + Z_{0e}}{3Z_{0o} + Z_{0e}} Z_{0e}$$

$$Z_{o4} = \frac{Z_{0o} + Z_{0e}}{3Z_{0e} + Z_{0o}} Z_{0o}$$

The effective capacitance network for the two wire and four wire models are shown below—



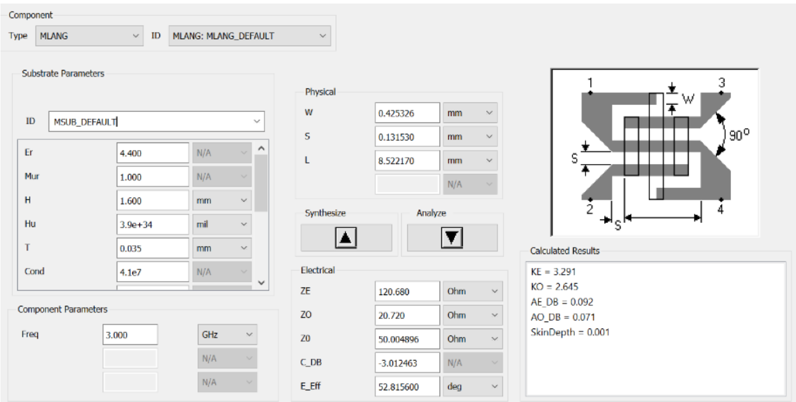


After these calculations, the length, width and gap between the lines can be calculated from ADS software using **LineCalc**.

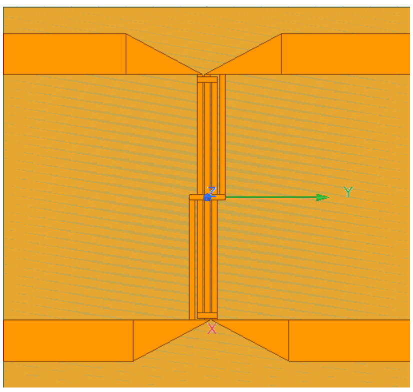
Calculations: A coupling factor of 0.707 is chosen in this design. The following python script is written for the required calculations—

```
import math

c = float(input("Coupling factor: "))
z0 = 50

z0e = (4*c - 3 + math.sqrt(9-8*c*c))/(2*c*math.sqrt((1-c)/(1+c)))*z0
z0o = (4*c + 3 - math.sqrt(9-8*c*c))/(2*c*math.sqrt((1+c)/(1-c)))*z0

ze4 = ((z0o+z0e)/(3*z0o+z0e))*z0e
zo4 = ((z0o+z0e)/(3*z0e+z0o))*z0o

print(ze4)
print(zo4)
```

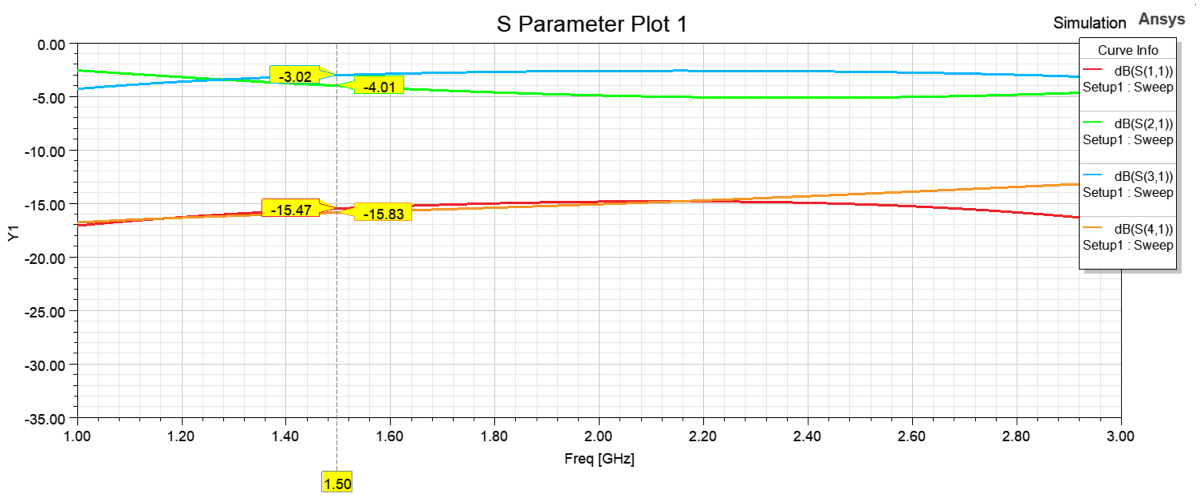
After getting the values of Z_{e4} and Z_{o4}, the length, width and gap of the lines are measured from the LineCalc—



Design and Results: The device is then designed in Ansys HFSS ®, and simulated at 1.5 GHz.The design is shown below

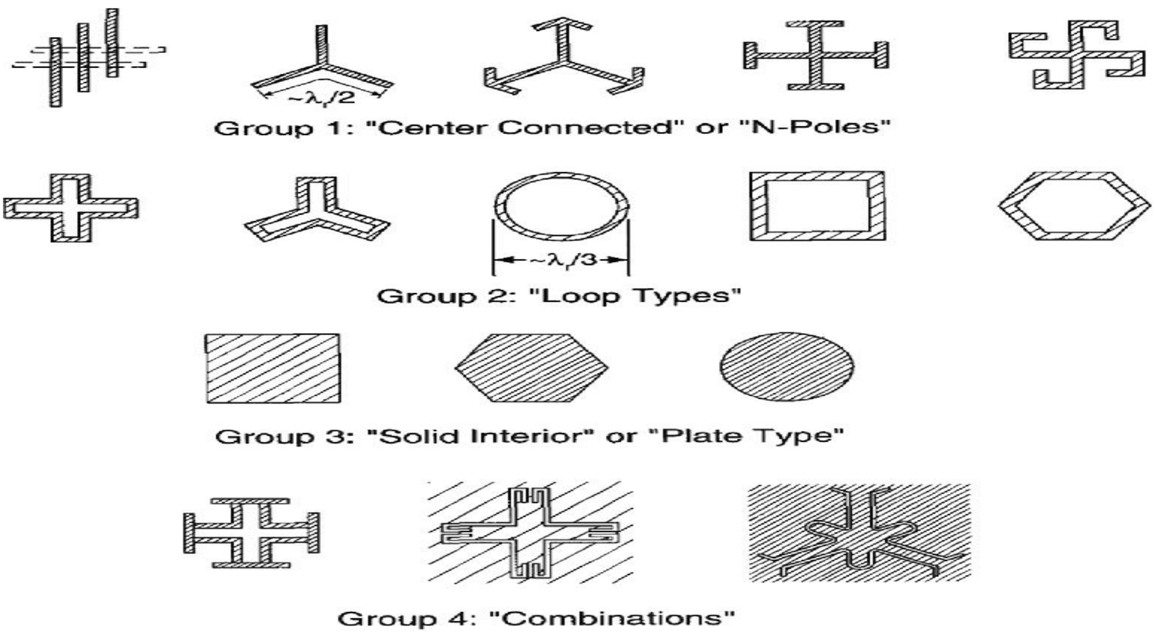


This design yields the below results –



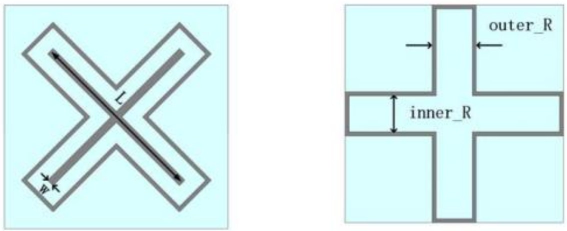
Experiment 8: Simulation of an FSS using HFSS

Introduction: A frequency selective surface is composite material designed to be transparent in some frequency bands while reflective, absorbing or redirecting to others. They are typically flat and composed of metal screen. It is an infinite array of a one-dimensional or two-dimensional periodic metal structure unit. It is widely used as a band-pass, band-stop spatial filter. A variety of military applications including design of antenna radomes, dichotic surfaces for reflectors and subreflectors of large aperture antennas, or even absorbers have used planar and curved FSSs.

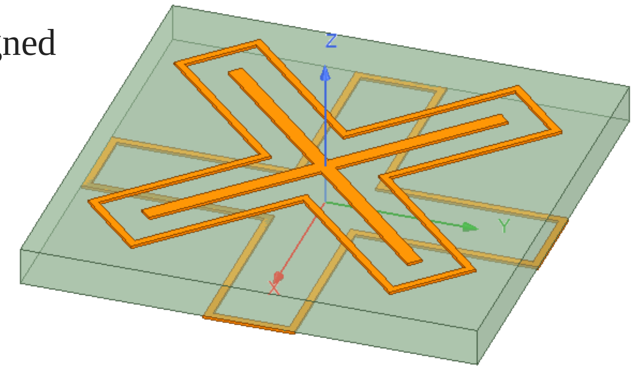


Several types of FSSs are shown in the above figure. These are nothing but periodic metal structures (only one unit of the infinite structure is shown in the above diagram). FSS are used widely as spatial band-pass, band-stop filter. Also in military applications like antenna radomes FSS are used.

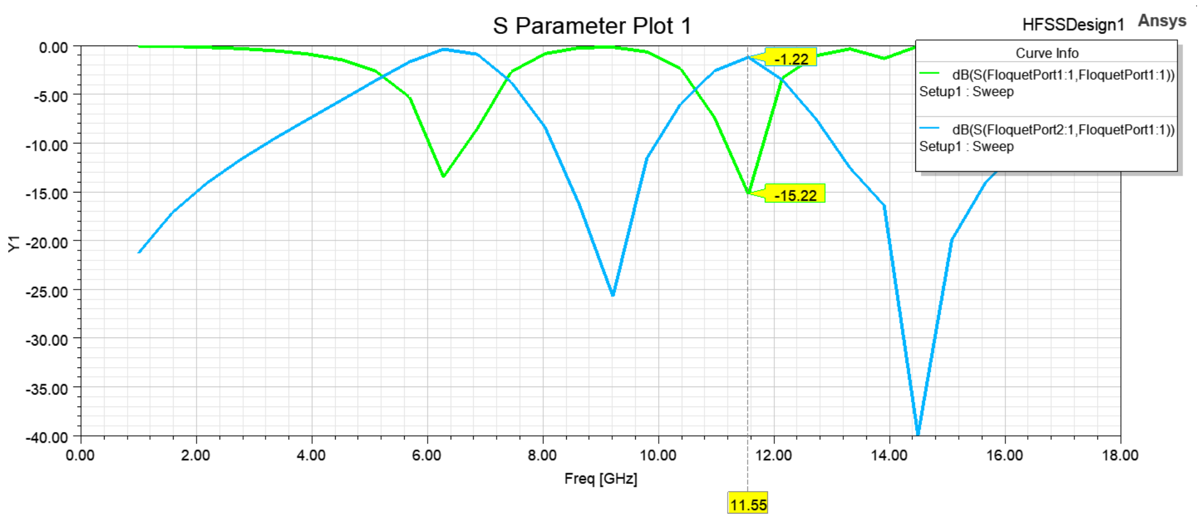
Design and Simulation: In this report, a FSS is designed and analyzed that consists of two cross-ring structures in the opposite sides. This FSS gives two bands of frequency response between 0 to 15 GHz that contains the radar frequency. The two sides of the FSS are shown in the diagram beside. This is one unit cell of the proposed structure. This unit cell is repeated infinite times to obtain the original FSS. There is also a cross-patch in the front side in the cross ring. The substrate height, length and width of the lines are optimized to obtain the desired centre frequencies.



The figure aside shows the unit cell designed in Ansys HFSS. A floquet port is defined to excite the structure as the floquet port is used to excite the unit cells to analyse the infinite structures.



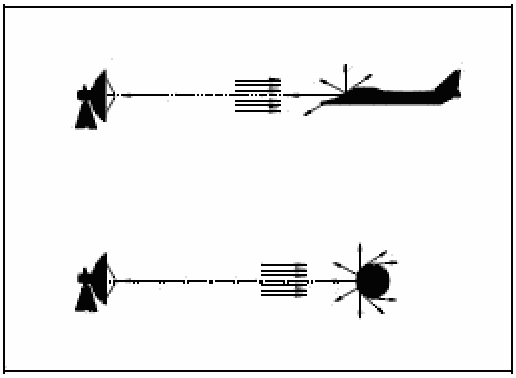
Results: The following curve shows the S_{11} and S_{21} characteristics of the FSS.



The green curve corresponds to the S_{11} parameter and blue curve shows the S_{21} parameter. Two centre frequencies are obtained at 11.5 GHz and 6.3 GHz.

Experiment 9: Radar Cross Section of ans Antenna

Introduction: Radar Cross Section or RCS of any object is a measure of how much it is detectable by a Radar. It is a measure of the ratio of backscatter power per steradian in the direction of the radar to the power density that is intercepted by the target. For this measurement, the reference is taken as the power reflected by an ideal reflecting sphere of cross section area 1 m^2. The ratio of the power scattered by any object to the power scattered by the sphere is taken to calculate the RCS of the mentioned object. The figure aside shows the process followed to calculate the RCS of an aircraft. RCS of any object depends on three factors –

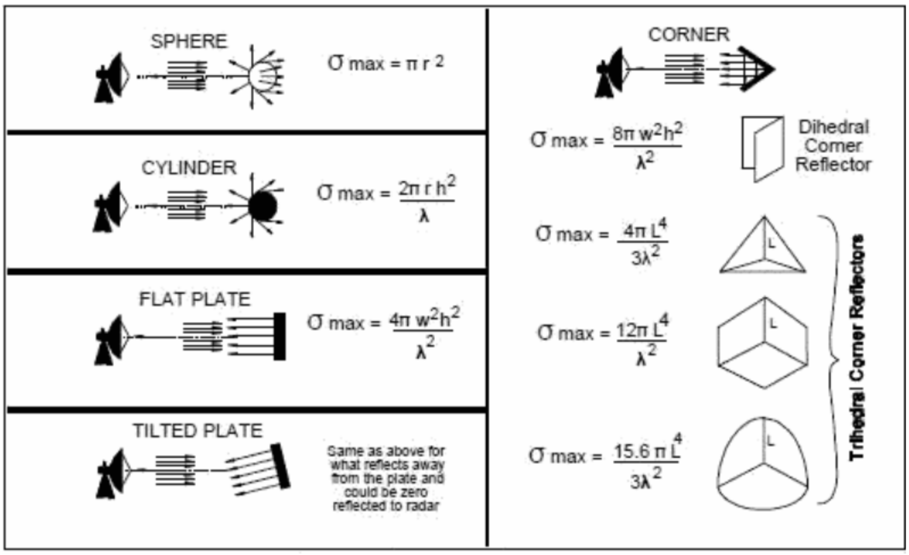


1. Projected Cross section
2. Directivity
3. Reflectivity

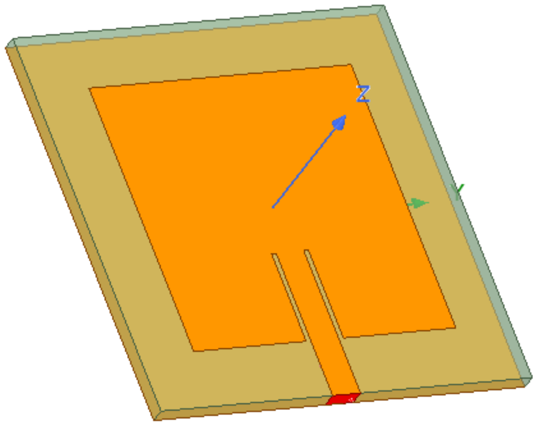
The first one is the area experienced by the transmitted power by the Tx. Directivity is the measure of how directional is the signal reflected by the object and Reflectivity signifies the amount of power reflected by the object.

$$RCS(\sigma) = Projected\ Cross\ Section\ \times Directivity\ \times Reflectivity$$

RCS for some objects is shown below –



Simulation and Result: A patch antenna is designed to do RCS analysis in Ansys HFSS. It radiates at a centre frequency of 2.4 GHz. The antenna is shown below –



A plane wave source is defined to be incident on the antenna. After that, the Bistatic RCS plot in Azimuthal plane is shown as below –