Micro-Patch Antenna using MATLAB

OBJECTIVE :

To design a Micropatch antenna using MATLAB on printed circuit board design.

ABSTRACT :

This project focuses on the design and simulation of a compact antenna system integrated with a printed circuit board (PCB). The dimensions of the PCB, including its thickness, length, and width, are specified along with the material properties, such as the dielectric constant (ε_r) of FR4 material. The antenna design includes a feed plane and ground plane, constructed using rectangular geometries. Various shapes of antennas are created, including rectangles of different dimensions, to achieve desired radiation characteristics.

The PCB stack is constructed, incorporating the antenna structures and dielectric material. Simulations are performed to analyze the radiation pattern and impedance characteristics of the antenna system across a frequency range. The S-parameter analysis is conducted to evaluate the reflection coefficient (S11) of the antenna design.

Furthermore, Gerber files are generated for fabrication purposes, ensuring the practical implementation of the designed antenna system. This project provides insights into the design and simulation of compact antennas integrated with PCBs, catering to diverse applications in wireless communication systems.

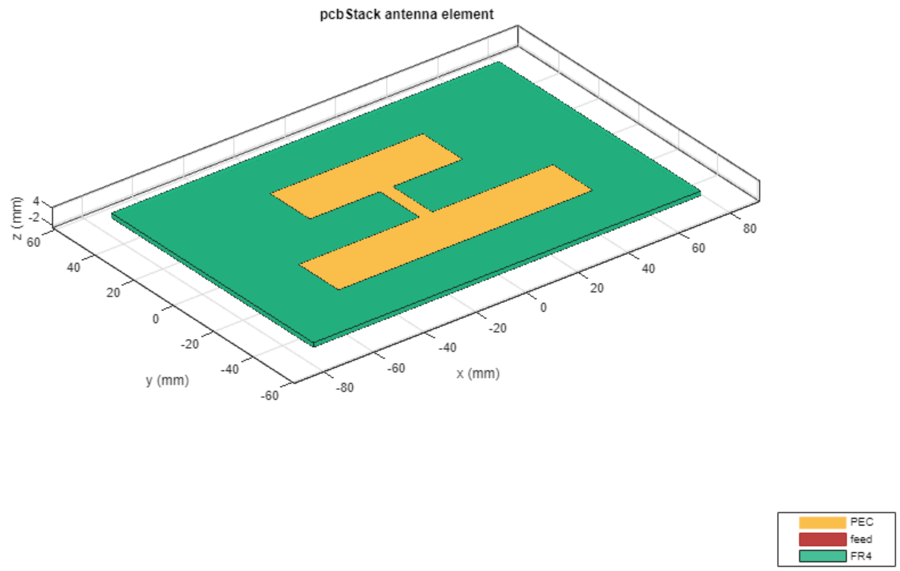
INTRODUCTION:

In telecommunication, A microstrip or micropatch antenna (also known as a printed antenna) usually means an antenna fabricated using microstrip techniques on a printed circuit board (PCB). An individual microstrip antenna consists of a patch of metal foil of various shapes (a patch antenna) on the surface of a PCB (printed circuit board), with a metal foil ground plane on the other side of the board. The antenna is usually connected to the transmitter or receiver through foil microstrip transmission lines. The radiofrequency current is applied (or in receiving antennas the received signal is produced) between the antenna and ground plane.

HARDWARE/SOFTWARE REQUIREMENT & DESCRIPTION:

MATLAB software

BLOCK DIAGRAM/ INTERFACE DIAGRAM:



REALISTIC CONSTRAINTS:

Realistic constraints for this project include managing costs and adhering to deadlines. Designs must be manufacturable, meet performance requirements, and be compatible with standard processes and regulations. Additionally, reliability and physical constraints such as size, weight, and environmental factors need consideration.

APPROACH/METHODOLOGY:

- Specify PCB dimensions (1.6mm thickness, 152.4mm length, 101.6mm width) and FR4 material (ε_r=4.4).
- Create dielectric layer and design antenna geometries (e.g., rectangles).
- Construct PCB stack integrating antennas with ground and feed planes.
- Simulate radiation patterns, impedance, and S-parameters across frequency range.
- Generate Gerber files for fabrication.
- Consider constraints: cost, time, manufacturability, performance, regulations, reliability, and physical dimensions.

RESULT: Micropatch antenna was successfully designed in MATLAB and greb file was exported

CONCLUSION: In conclusion, this project successfully designs and simulates a compact antenna system integrated with a printed circuit board (PCB). By optimizing dimensions, material selection, and antenna geometries, the system demonstrates efficient radiation patterns and impedance characteristics across a specified frequency range, paving the way for practical implementation in wireless communication applications.

REFERENCES:

https://in.mathworks.com/matlabcentral/fileexchange/74724-designing-micropatch-antenna-using-matlab
https://matlab.mathworks.com/

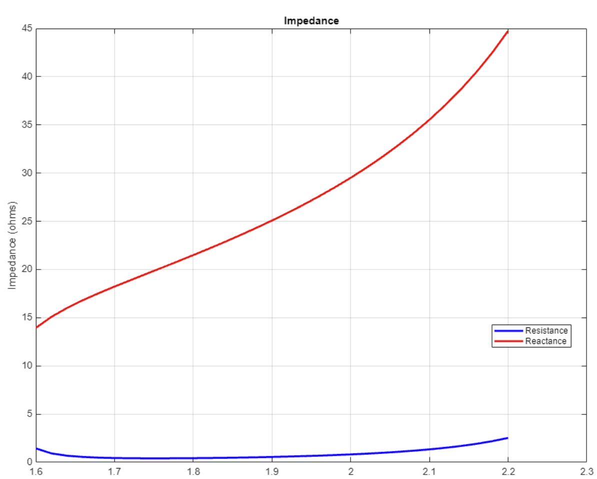
APPENDIX:

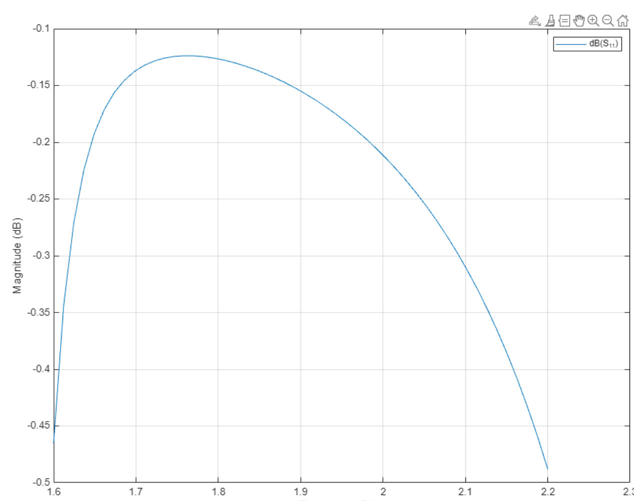
MATLAB Code:
Script file containing the MATLAB code used for the design, simulation, and analysis of the antenna system integrated with the PCB.

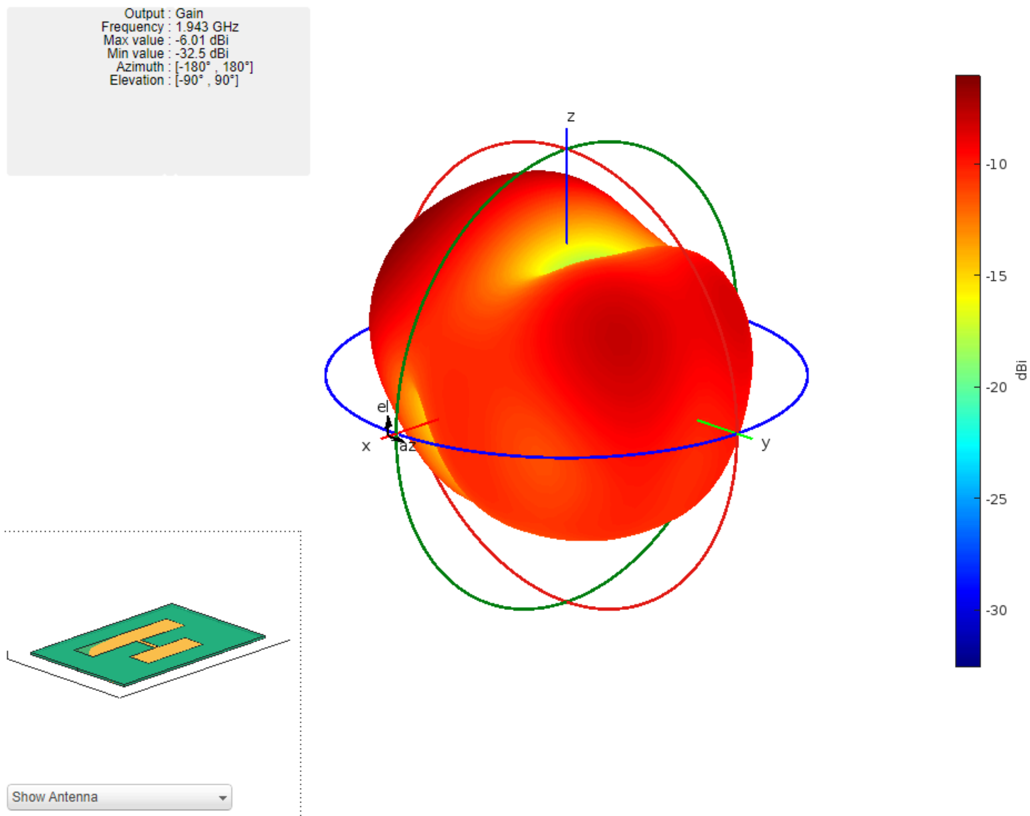
```
%Size of PCB
pcbThickness = 1.6e-3;  %1.6mm
pcbLength = 152.4e-3;   %152.4mm or 6inch
pcbWidth = 101.6e-3;    %101.6mm 0r 4inch

%Specifying Material of PCB
pcbMaterial = 'FR4';
pcbEpsilonR = 4.4;

%Creating dielectic Material
d = dielectric(pcbMaterial);
d.EpsilonR = pcbEpsilonR;
d.Thickness = pcbThickness;

AntennaPlane=antenna.Rectangle('Length',0.5e-2,'Width',5e-2,'Center',[0, 0]); %Creating Feed Plane of Antenna
GndPlane = antenna.Rectangle('Length',pcbLength,'Width',pcbWidth); %Creating Ground Plane of Antenna

%Creating Different Shapes of antenna
Rec = antenna.Rectangle('Length',10e-2,'Width',2e-2,'Center',[0,-20e-3]);
Rec1 = antenna.Rectangle('Length',6e-2,'Width',2e-2,'Center',[0,20e-3]);

%%Creating PCB Stack
p = pcbStack;
p.Name = 'Strip-fed slot';
p.BoardShape = GndPlane;
p.BoardThickness = pcbThickness;
p.Layers = {AntennaPlane,d,GndPlane}; %[x Cordinate,y Cordinate,startLayer stopLayer]
p.FeedLocations = [0,(-pcbWidth/2)+6e-2,1,3];

%Adding all different shapes of antenna
AntennaPlane = AntennaPlane + Rec + Rec1;
p.Layers = {AntennaPlane,d,GndPlane};

%Plotting Different patterns and graphs
```

```
figure(1);
show(p); %Display Antenna

figure(2);
pattern(p,1.943e9); %Display Radiation Pattern at 1.943GHZ

figure(3);
impedance(p,1.6e9:2e7:2.2e9); %Display Impedance Graph from 1.6GHz to 2.2GHz

freq = linspace(1.6e9, 2.2e9, 50);  % Creating Frequency Vector
s = sparameters(p,freq,50);     % Calalculate S11 for all frequencys

figure(4);
rfplot(s);%Diplay S11 Plot


%Generating Gerber Files for Fabrication
C = PCBConnectors.SMA_Cinch;
W = PCBServices.PCBWayWriter;
W.Filename = 'antenna_design_file';

gerberWrite(p,W,C);

%This will genrate a ZIP file in project folder with Name "antenna_design_file.zip"
```

Simulation Results:
Plots and graphs illustrating radiation patterns, impedance characteristics, and S-parameter analysis across the frequency range.







Output : Gain
Frequency : 1.943 GHz
Max value : -6.01 dBi
Min value : -32.5 dBi
Azimuth : [-180° , 180°]
Elevation : [-90° , 90°]
z
x
y
el
az
-10
-15
-20
-25
-30
dBi
Show Antenna