WAVE PROPAGATION AND ANTENNA

PROJECT REPORT

PREPARED BY:
HUZAIFA ZAFAR
MAIRA WAHEED
HAJRA BIBI

PREPARED FOR:
DR. QASIM

NUST
CEME

Title

Design and Implementation of Microstrip Inset Feedline Square Patch Antenna

Project Statement

Q: Design a microstrip square patch antenna of 2.4 GHz using FR4 as a substrate/dielectric. Minimum Bandwidth is 50 MHz and a gain of 4-5 dBi. A microstrip feedline topology should be used as a feeding mechanism. HFSS Simulation should be included. PCB Fabrication should be performed using etching. The height of the dielectric is 1.6 mm.

Objectives

- Design a microstrip square patch antenna operating at 2.4 GHz frequency.

- Utilize FR4 as the substrate/dielectric material for the antenna.

- Ensure the antenna achieves a minimum bandwidth of 50 MHz.

- Aim for a gain of 4-5 dBi for the designed antenna.

- Implement a microstrip feedline topology for the feeding mechanism.

- Include HFSS simulation to validate the design and performance of the antenna.

- Perform PCB fabrication using etching techniques.

- Maintain a dielectric height of 1.6 mm throughout the design process.

Procedure

- Calculate dimensions for the square patch antenna for 2.4 GHz frequency and FR4 substrate (1.6 mm height).

- Design microstrip feedline for impedance matching with the antenna.

- Simulate antenna performance using HFSS software.

- Transfer antenna design to PCB layout software.

- Fabricate PCB using etching techniques on FR4 substrate.

- Mount components and connectors onto the fabricated PCB.

- Test antenna performance for parameters like return loss, bandwidth, and gain.

- Iterate design if necessary for optimization.

- Document entire process including calculations, simulations, fabrication, testing, and any optimizations made.

- Prepare a comprehensive report detailing all steps and results.

Calculations

Width:

$$W = \frac{1}{2f_r\sqrt{(u_0\,\varepsilon_0\,)}} \cdot \sqrt{\left(\frac{2}{\varepsilon_r + 1}\right)}$$

Where:

f_r = 2.4 GHz , u_0 = 1.25663706 × 10^{-6} m*kg /s^2A^2, ε_0 = 8.8541878128(13)×10^{-12} F·m^{-1}, ε_r = 4.4

Therefore:

W = 0.03803629872 cm = 38.03628872 mm

Effective Permittivity:

$$\varepsilon_{eff} = \frac{(\varepsilon_r + 1)}{2} + \frac{(\varepsilon_r - 1)}{2}\left(1 + \frac{12h}{w}\right)^{-\frac{1}{2}}$$

Where:

$$\boldsymbol{\varepsilon_r = 4.4\,, h = 1.6\, mm}$$

Therefore:

$$\boldsymbol{\varepsilon\mathrm{eff} = 4.085837332}$$

Fringing Effect:

$$\boldsymbol{\Delta(L) = 0.412h \times \frac{(\varepsilon_{\mathrm{eff}} + 0.3) \times \left(\frac{W}{h} + 0.264\right)}{(W - 0.258) \times \left(\frac{W}{h} + 0.8\right)}}$$

Therefore:

= $7.388192488 * 10^{-4}$ m = 0.7388192488mm

Wavelength:

$$\boldsymbol{\lambda = \frac{c}{f}}$$

Where:

c = $3*10^{8}$ m/s **, f =** 2.4 GHz

Therefore:

= 0.125m

$\frac{\lambda}{2}$ = 0.0625 m

Note: Length and Width of the substrate must be less than $\frac{\lambda}{2}$

Effective Length of Substrate:

$$L_{\text{eff}} = L + 2\Delta(L)$$

Where:

$L\ (length\ of\ substrate) = \frac{c}{2f}$ = 0.0625m = 62.5mm

Therefore:

= 0.0639776385 = 63.9776385 mm

Absolute Frequency Modes:

$$(f_r)_{010} = \frac{c}{2L\sqrt{e_r}}$$

= 1.14 * 10^9 Hz

$$(f_{rc})_{010} = \frac{1}{2L_{\text{eff}\sqrt{e_{\text{eff}}\mu_0\varepsilon_0}}}$$

= 1.16 * 10^9 Hz

Length of Square Patch:

$$L = \frac{c}{2f_{r\sqrt{e_{\text{eff}}}}} - 2\Delta(L)$$

Where:

= 0.02944236122 = 29.44236122 mm

R_{in}:

$$R_{\text{in}} = 90 \times \frac{e_r^2}{e_r - 1} \times \sqrt{\frac{L}{W}}$$

= 450.8750423 Ω

Z_{in}:

$$Z_c = \frac{60}{\sqrt{e_r}} \ln\left(\frac{8h}{W} + \frac{W}{4h}\right)$$

$$= 52.55449804\ \Omega$$

Depth of Inset Feed Cut:

$$D = \frac{L}{\pi} \cos^{-1}\left(\sqrt[4]{\frac{Z_{\text{feedline}}}{Z_{\text{antenna}}}}\right)$$

$$= 6.0706\ \text{mm}$$

Spacing of Cuts:

The initial value for the cuts was 3mm on both sides to make a total of 6 mm. However, we got better results for 3.1 mm on both sides or 6.2mm cuts. So we have used that in our model and our project.

HFSS Simulation

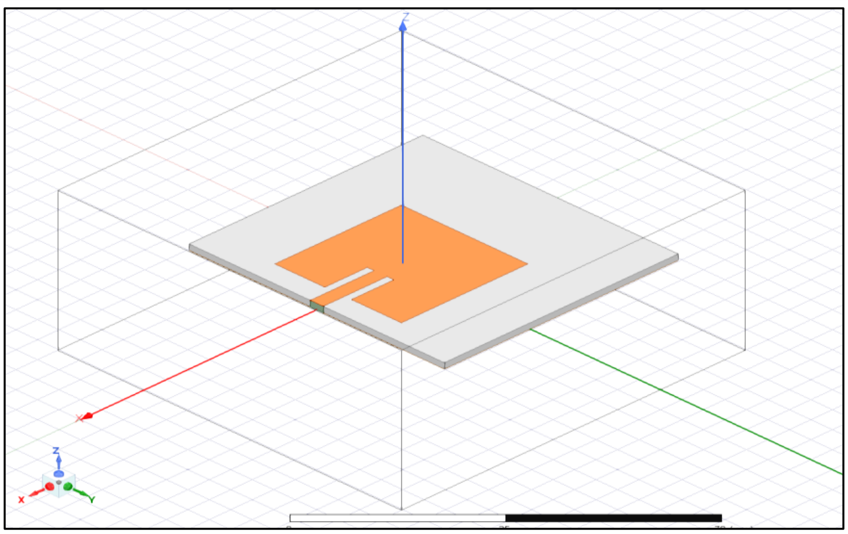


Figure 1: HFSS Model of the Square Patch Antenna with Microstrip Inset Feedline

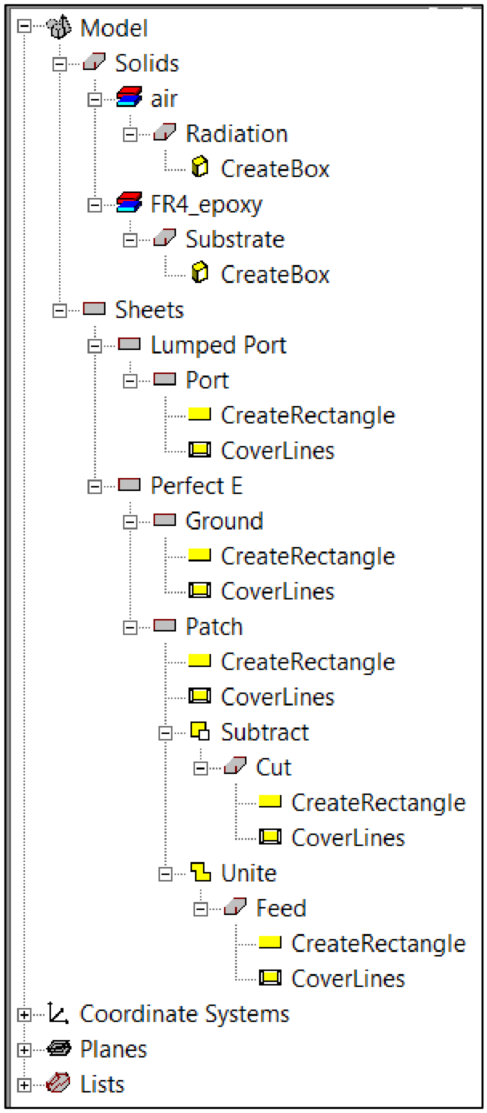
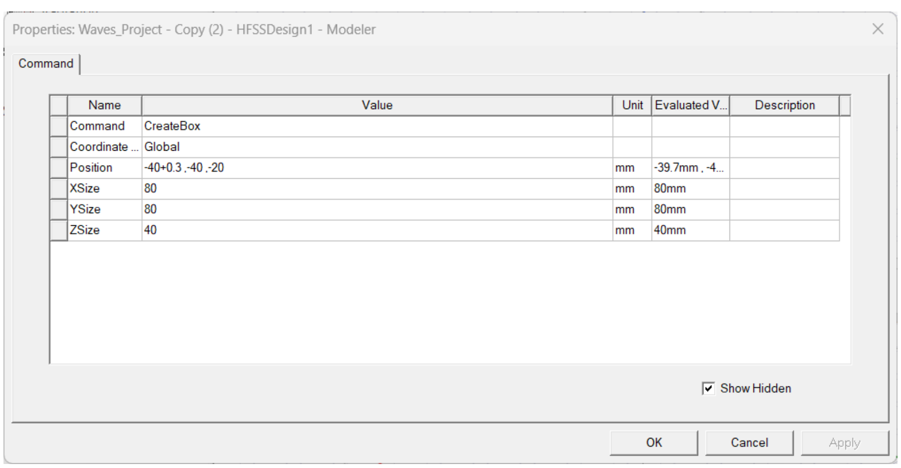
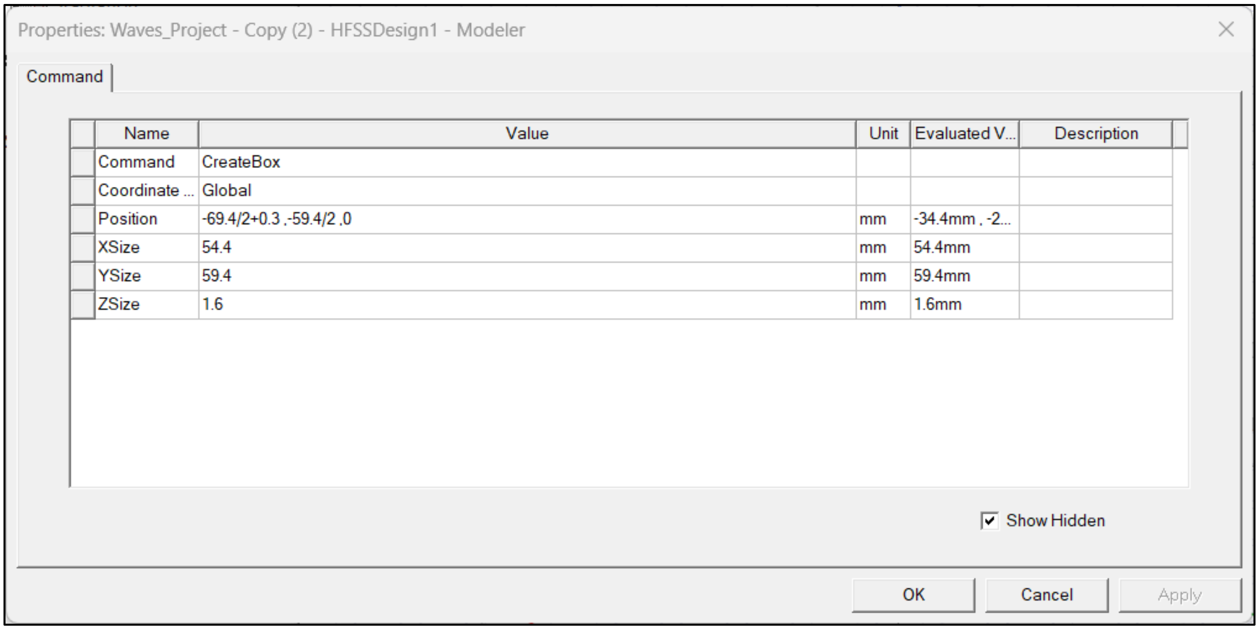


Figure 2: The components and materials that are used to make the model



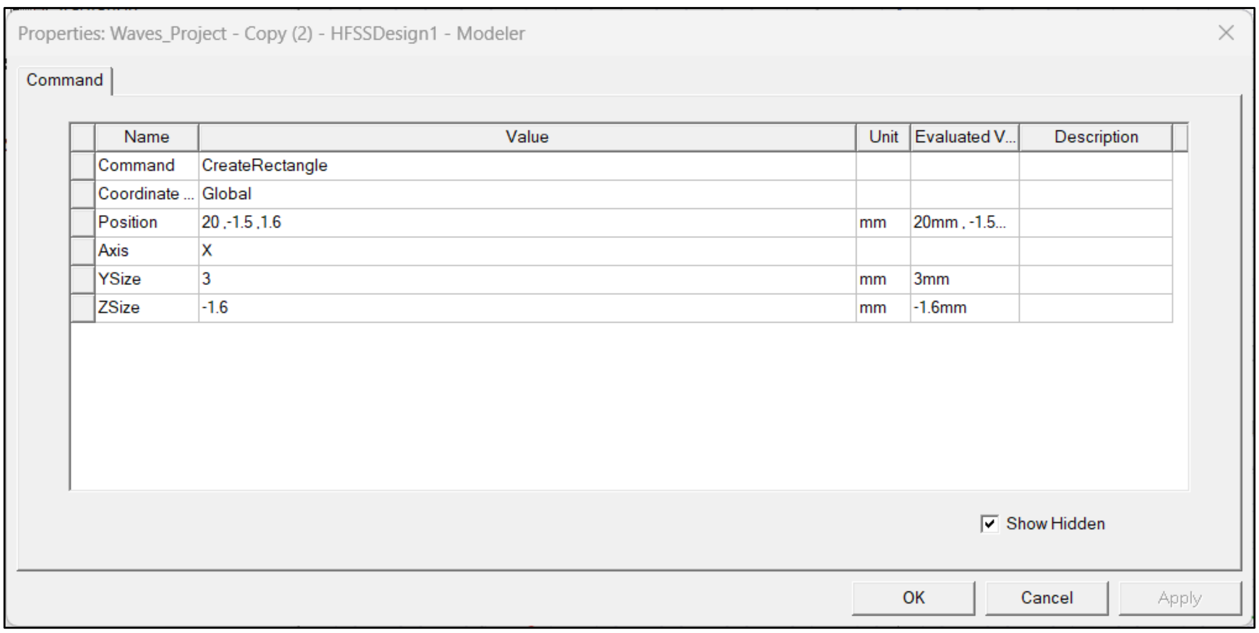
Name	Value	Unit	Evaluated V...	Description
Command	CreateBox			
Coordinate ...	Global			
Position	-40+0.3 ,-40 ,-20	mm	-39.7mm , -4...	
XSize	80	mm	80mm	
YSize	80	mm	80mm	
ZSize	40	mm	40mm	

Figure 3: Dimensions of Radiation Box



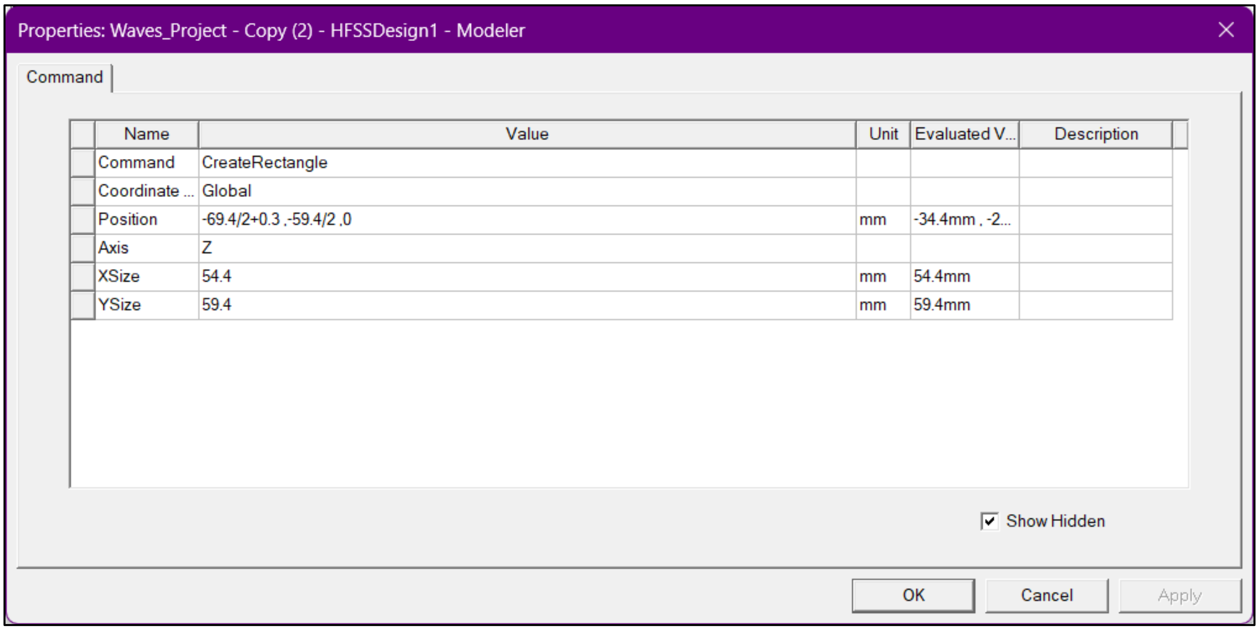
Name	Value	Unit	Evaluated V...	Description
Command	CreateBox			
Coordinate ...	Global			
Position	-69.4/2+0.3 ,-59.4/2 ,0	mm	-34.4mm , -2...	
XSize	54.4	mm	54.4mm	
YSize	59.4	mm	59.4mm	
ZSize	1.6	mm	1.6mm	

Figure 4: Dimensions of FR4 Substrate



Name	Value	Unit	Evaluated V...	Description
Command	CreateRectangle			
Coordinate ...	Global			
Position	20 ,-1.5 ,1.6	mm	20mm , -1.5...	
Axis	X			
YSize	3	mm	3mm	
ZSize	-1.6	mm	-1.6mm	

Figure 5: Dimensions of the Antenna Port


Properties: Waves_Project - Copy (2) - HFSSDesign1 - Modeler

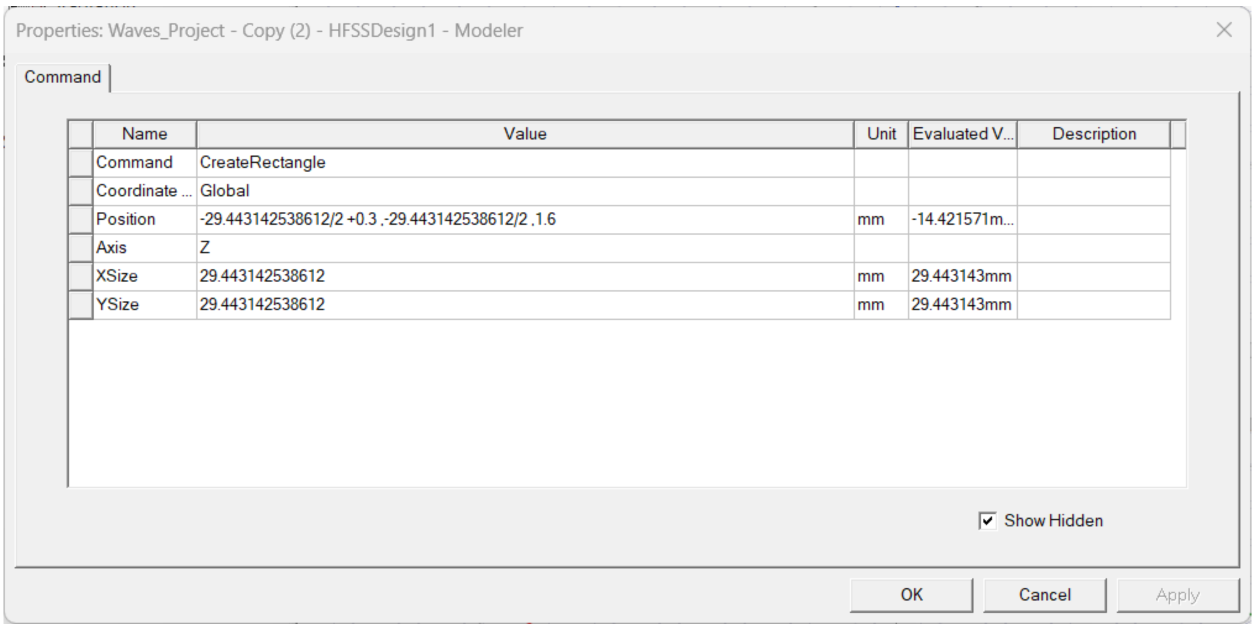
Command

Name	Value	Unit	Evaluated V...	Description
Command	CreateRectangle			
Coordinate ...	Global			
Position	-69.4/2+0.3 ,-59.4/2 ,0	mm	-34.4mm , -2...	
Axis	Z			
XSize	54.4	mm	54.4mm	
YSize	59.4	mm	59.4mm	

Show Hidden

OK Cancel Apply

Figure 6: Dimensions of the Patch Antenna Ground Plane


Properties: Waves_Project - Copy (2) - HFSSDesign1 - Modeler

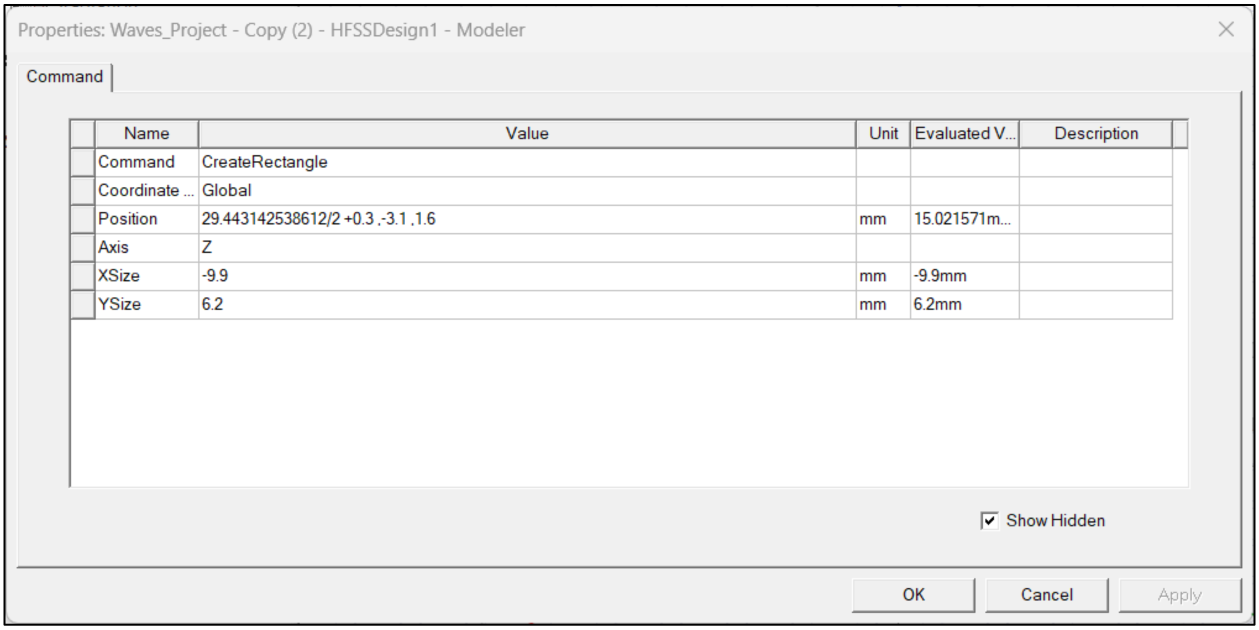
Command

Name	Value	Unit	Evaluated V...	Description
Command	CreateRectangle			
Coordinate ...	Global			
Position	-29.443142538612/2 +0.3 ,-29.443142538612/2 ,1.6	mm	-14.421571m...	
Axis	Z			
XSize	29.443142538612	mm	29.443143mm	
YSize	29.443142538612	mm	29.443143mm	

Show Hidden

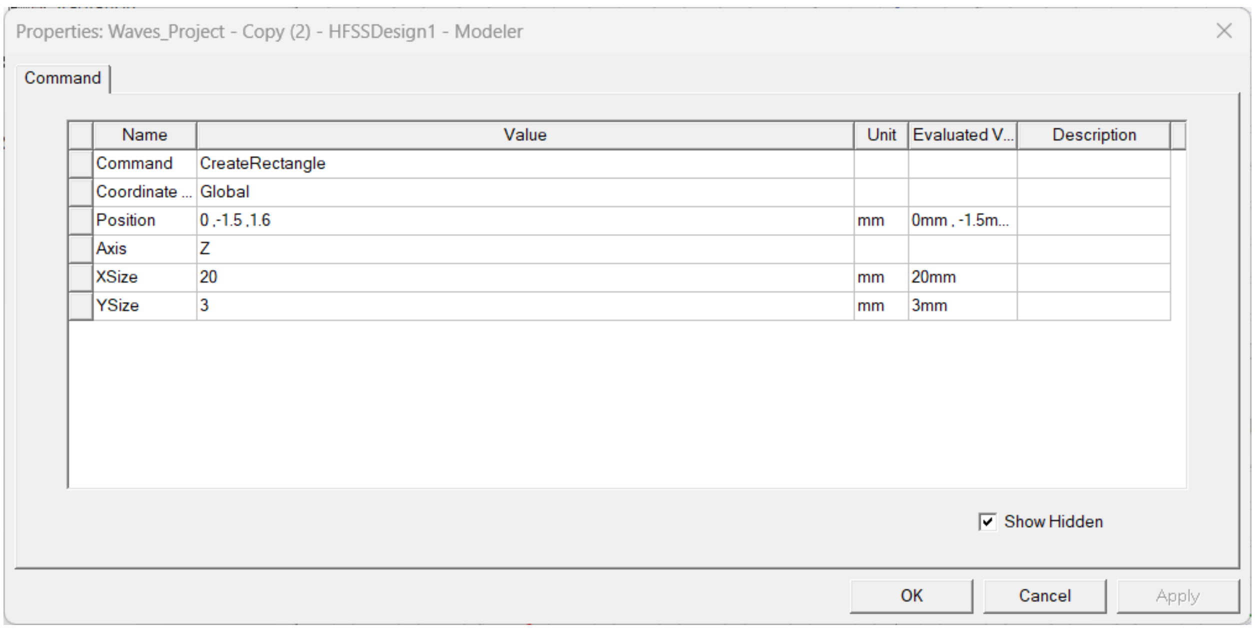
OK Cancel Apply

Figure 7: Dimensions of the Square Patch



Name	Value	Unit	Evaluated V...	Description
Command	CreateRectangle			
Coordinate ...	Global			
Position	29.443142538612/2 +0.3 ,-3.1 ,1.6	mm	15.021571m...	
Axis	Z			
XSize	-9.9	mm	-9.9mm	
YSize	6.2	mm	6.2mm	

Figure 8: Dimensions of the Cut made for the inset feedline



Name	Value	Unit	Evaluated V...	Description
Command	CreateRectangle			
Coordinate ...	Global			
Position	0 ,-1.5 ,1.6	mm	0mm , -1.5m...	
Axis	Z			
XSize	20	mm	20mm	
YSize	3	mm	3mm	

Figure 9: Dimensions of the Microstrip Feedline

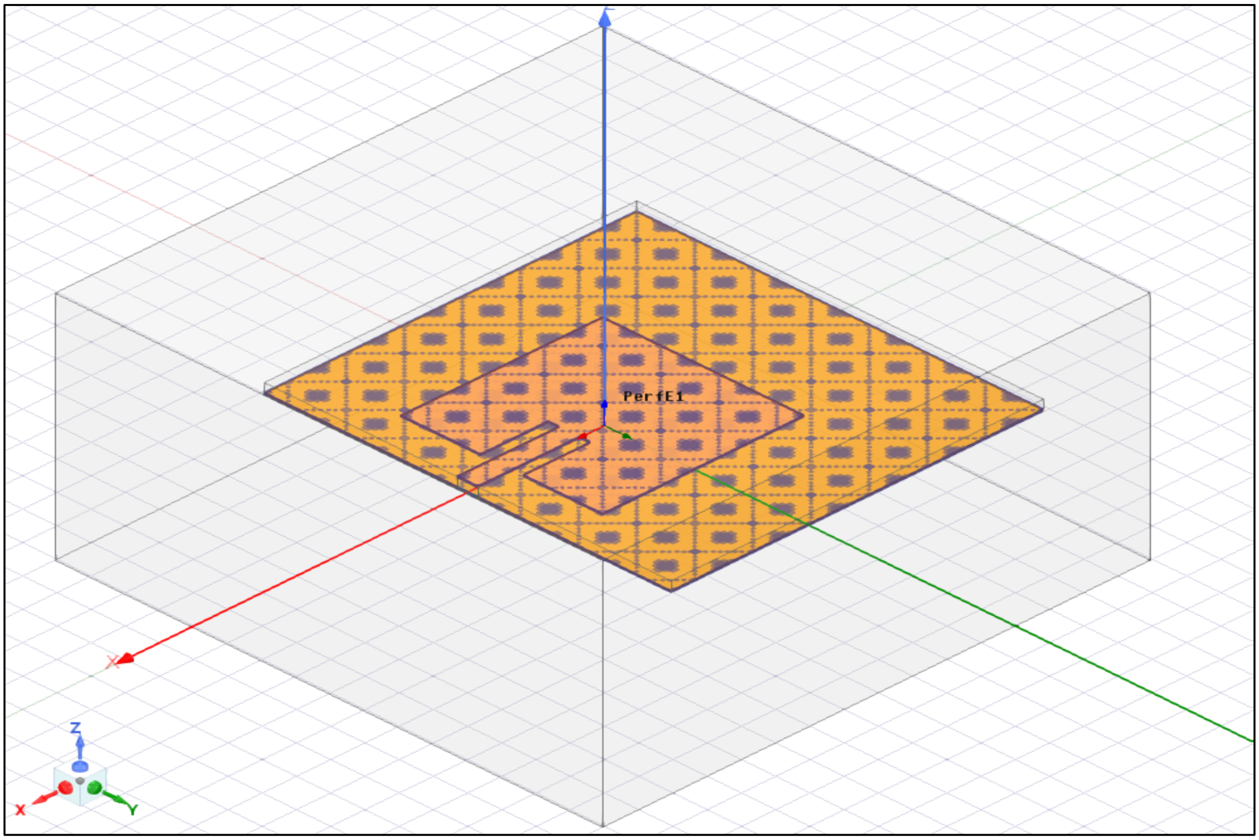


Figure 10: Perfect E Boundary on both the ground plane and the antenna patch

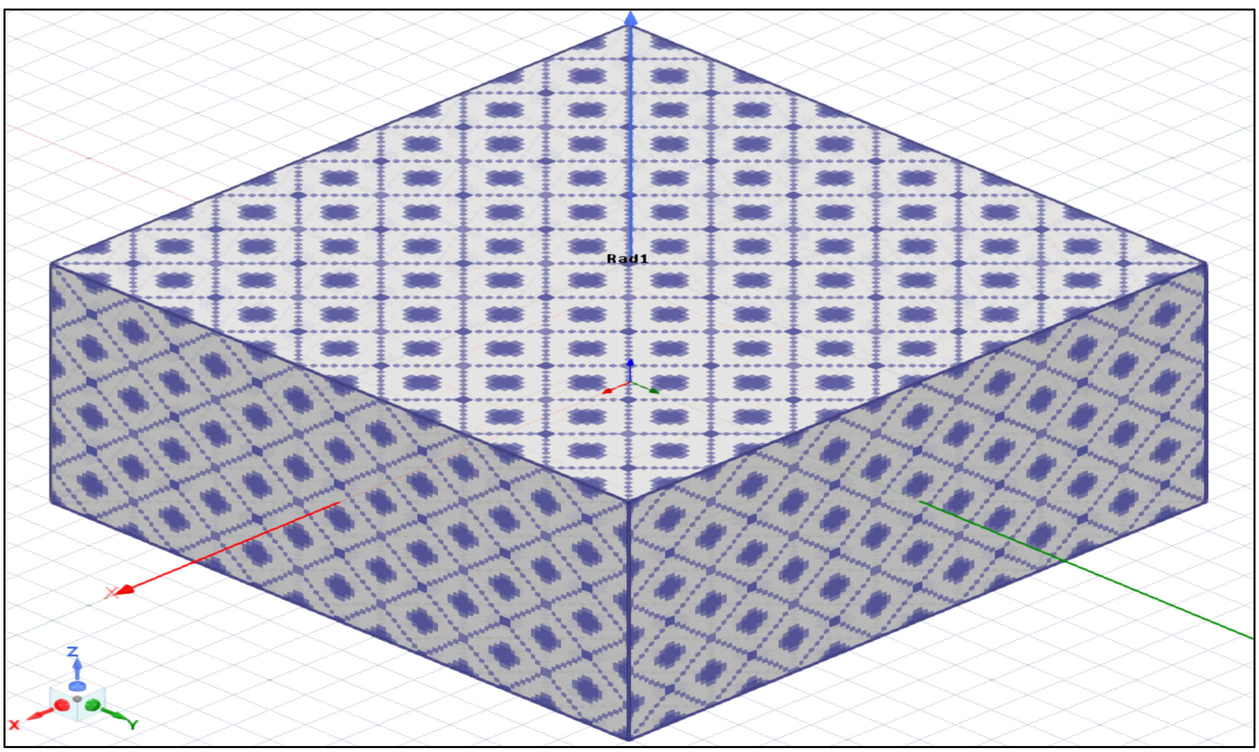


Figure 11: Radiation Boundary of the Antenna

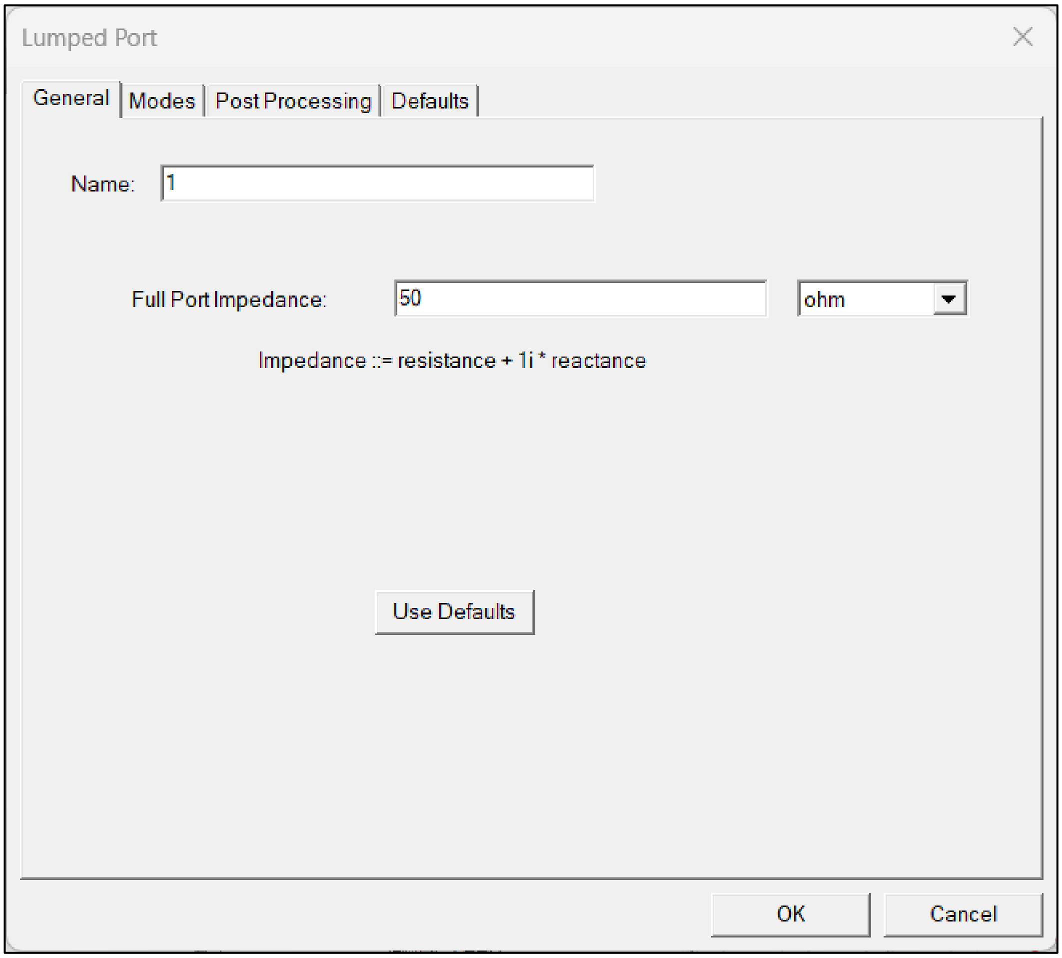


Figure 12: Patch Antenna Lumped Port Specifications

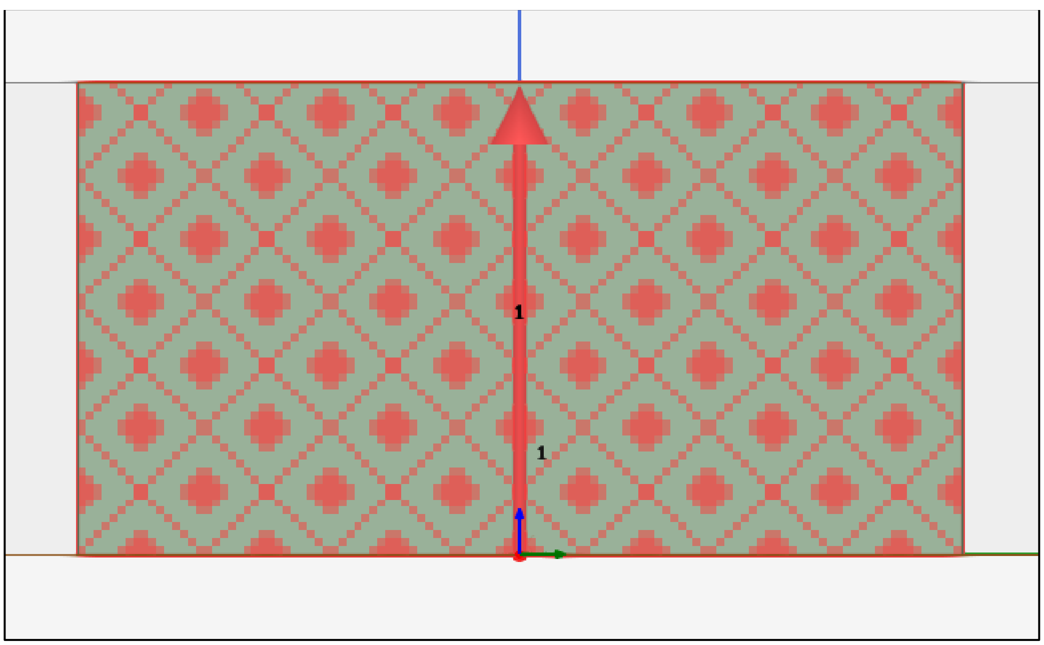


Figure 13: Port Excitation

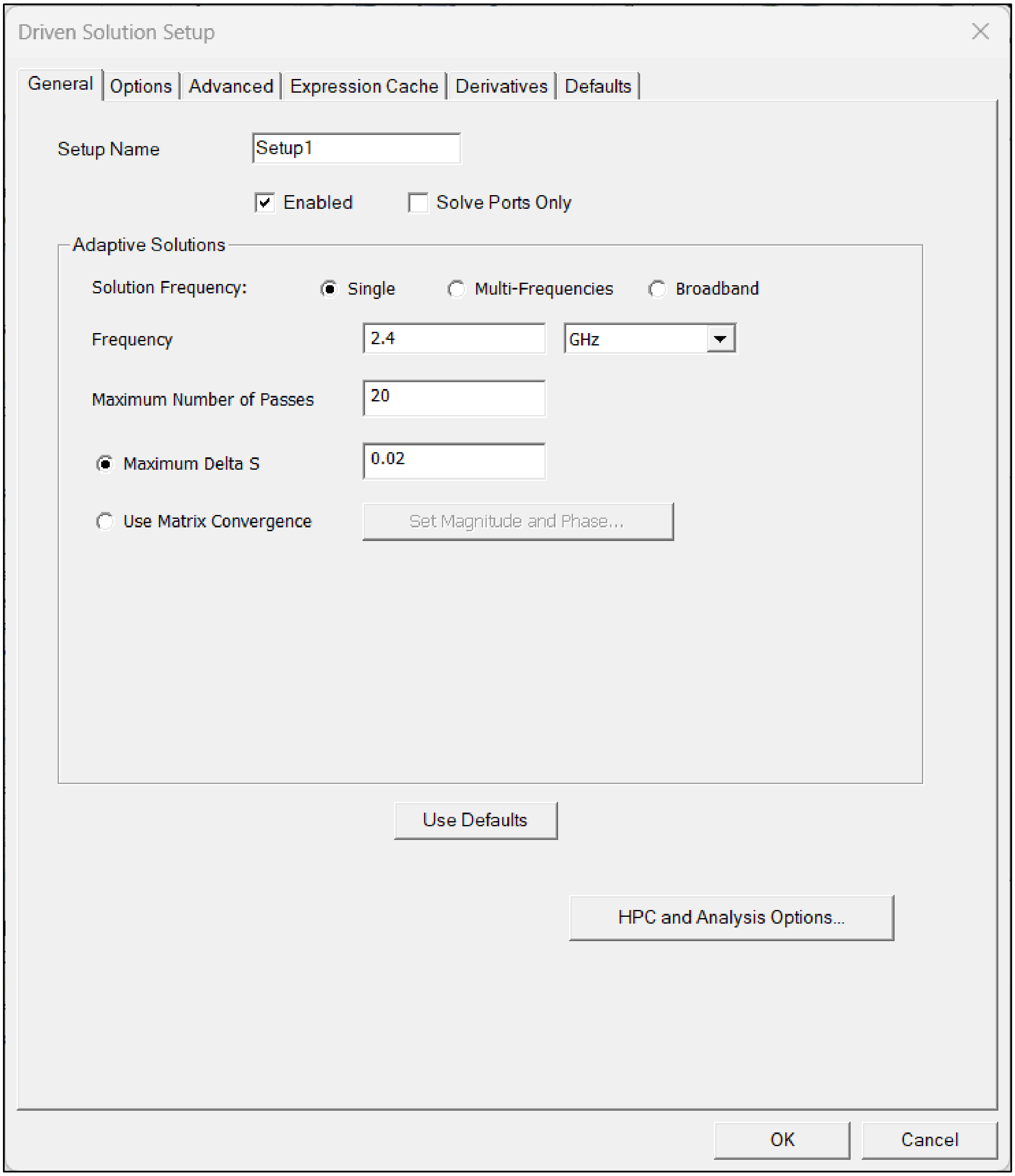


Figure 14: Advanced Setup Values

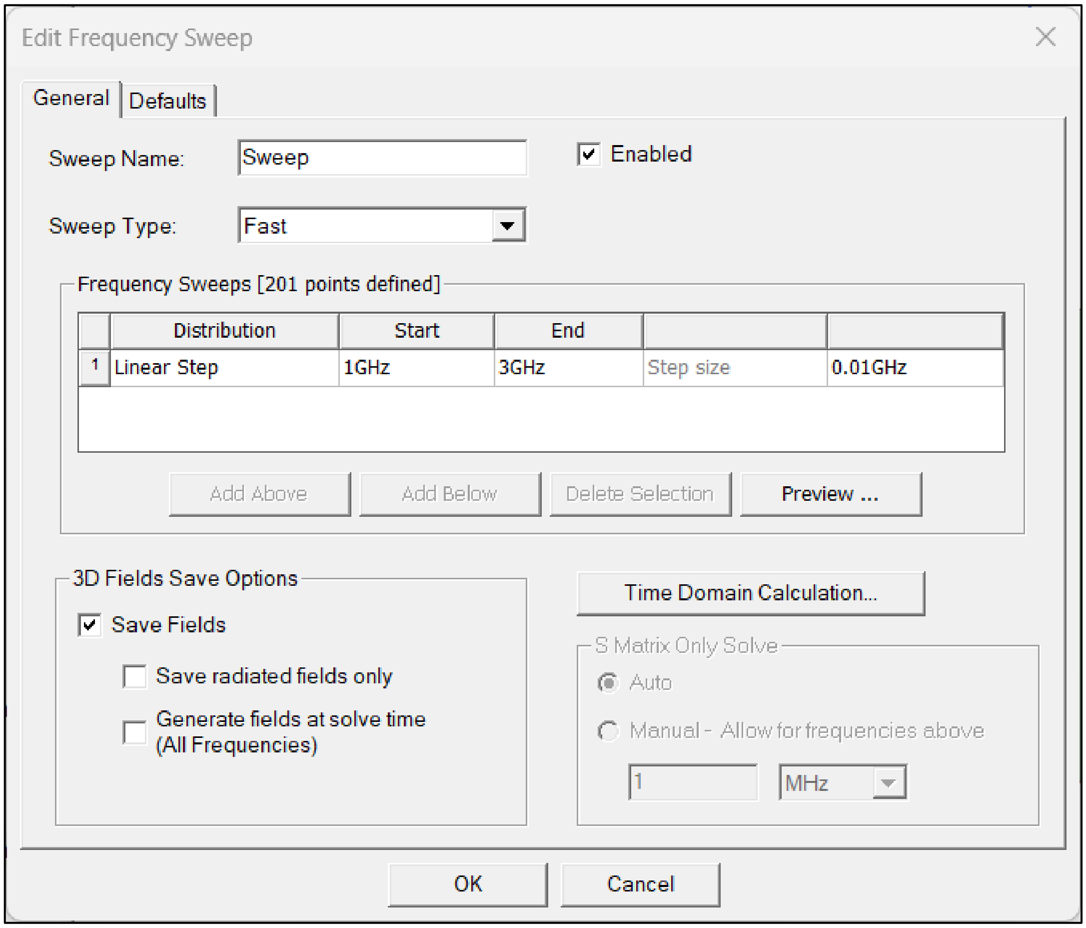


Figure 15: Frequency Sweep Settings

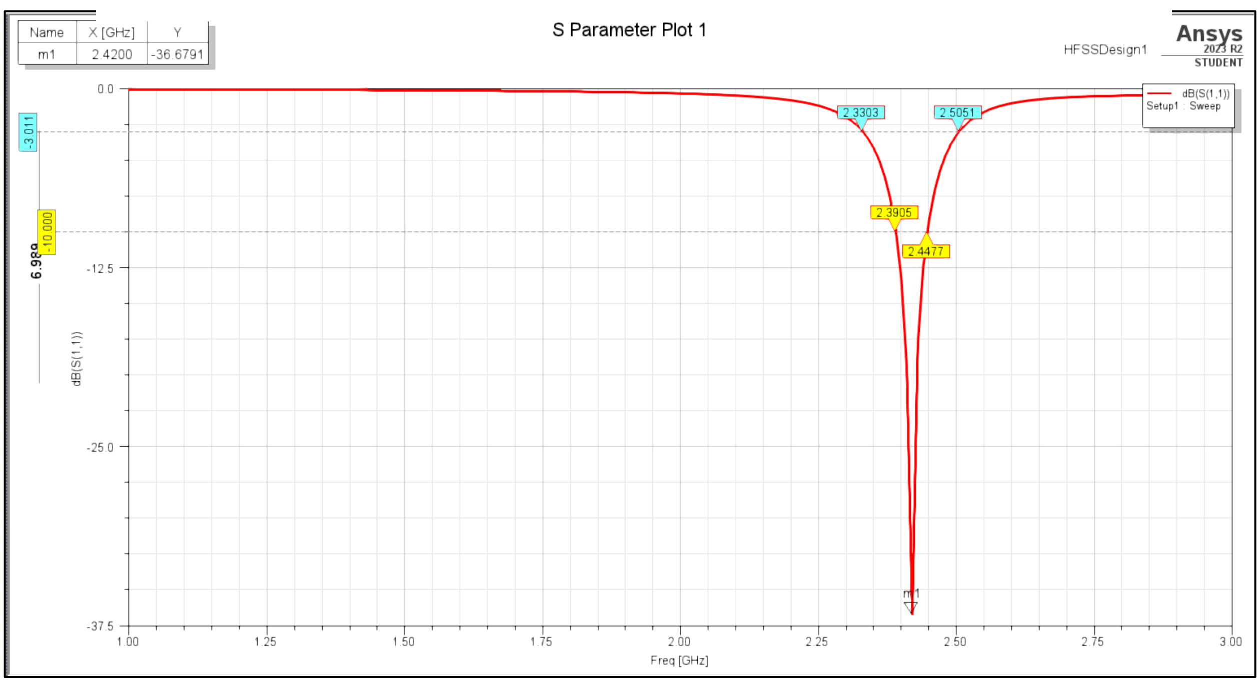


Figure 16: The S Parameter Plot of the Patch Antenna. It has a -36dB drop at 2.42 GHz. An almost 170 MHz Bandwidth at the -3 dB mark and at the -10 dB mark the bandwidth is almost 60 MHz.

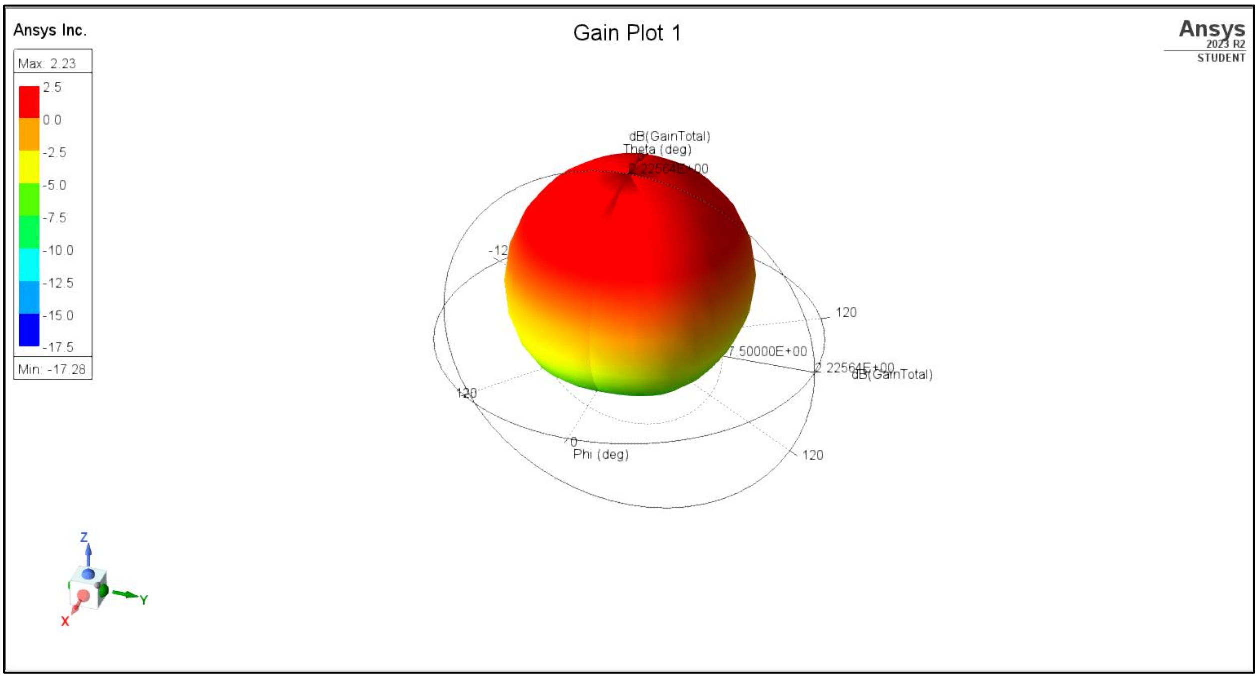


Figure 17: Far-Field Gain of the Antenna. 2.23 dB is the maximum value for gain.

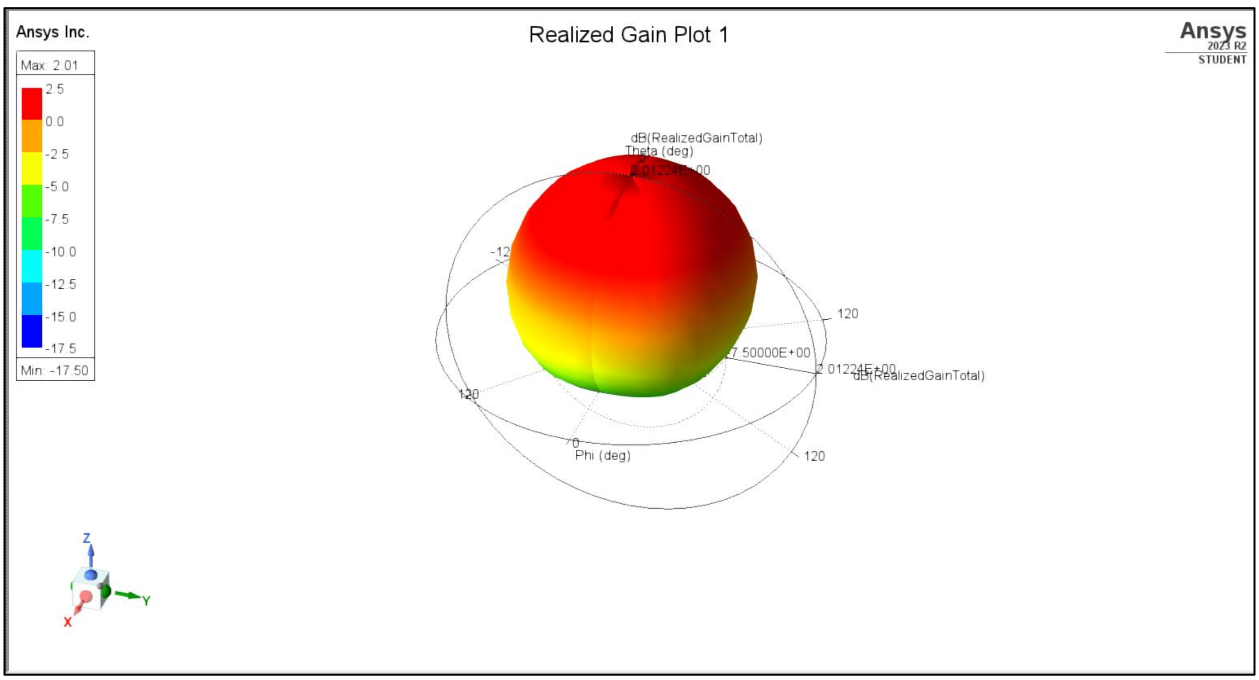


Figure 18: Realized Gain Plot of the Patch Antenna. 2.01 dB is the maximum value

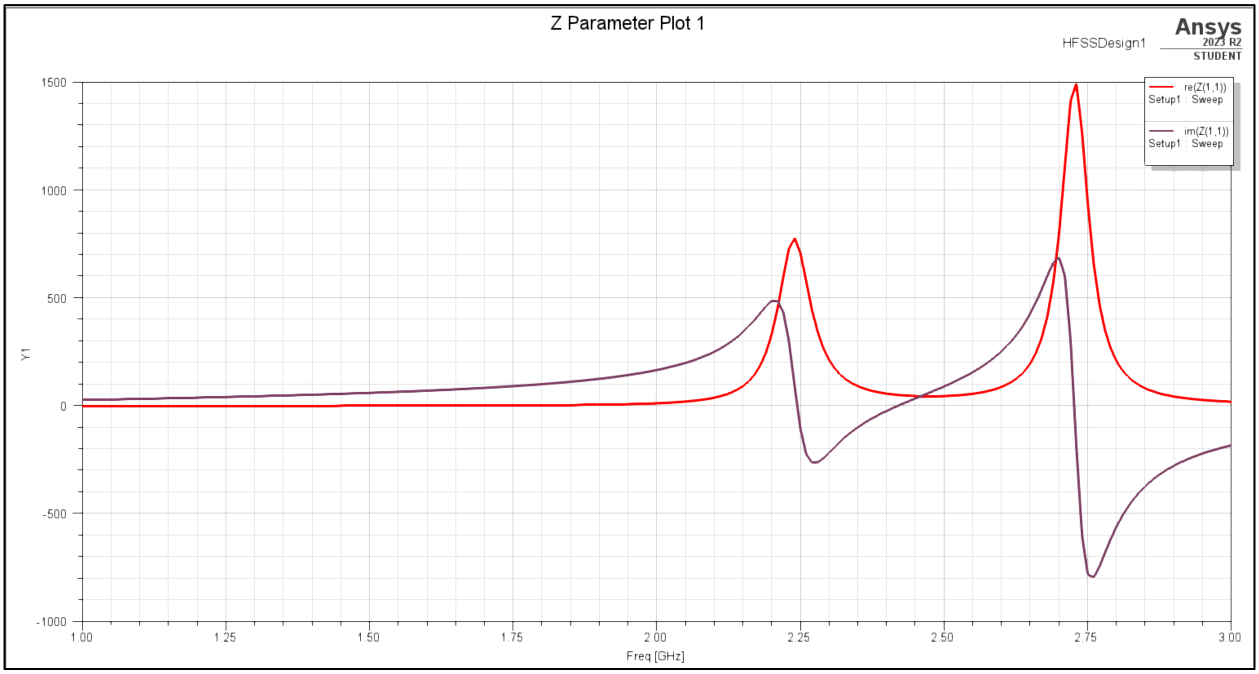


Figure 19: The Z Parameter Plot is shown. The real resistance(red) and the imaginary reactance (purple) can be seen. The reactance becomes zero at our resonant frequency which is a proof that we had done good matching of our antenna.

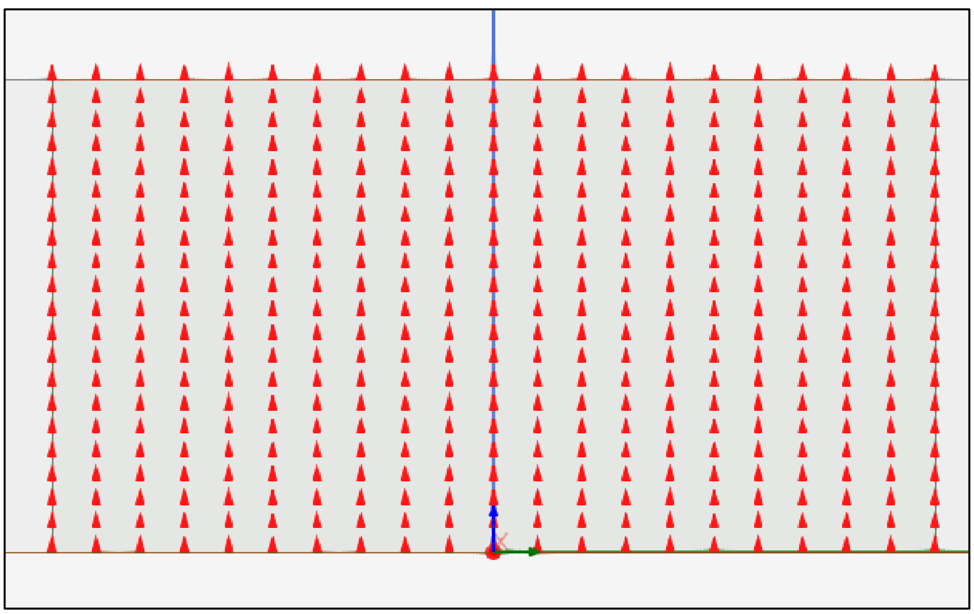


Figure 20: Mode of the Antenna Port

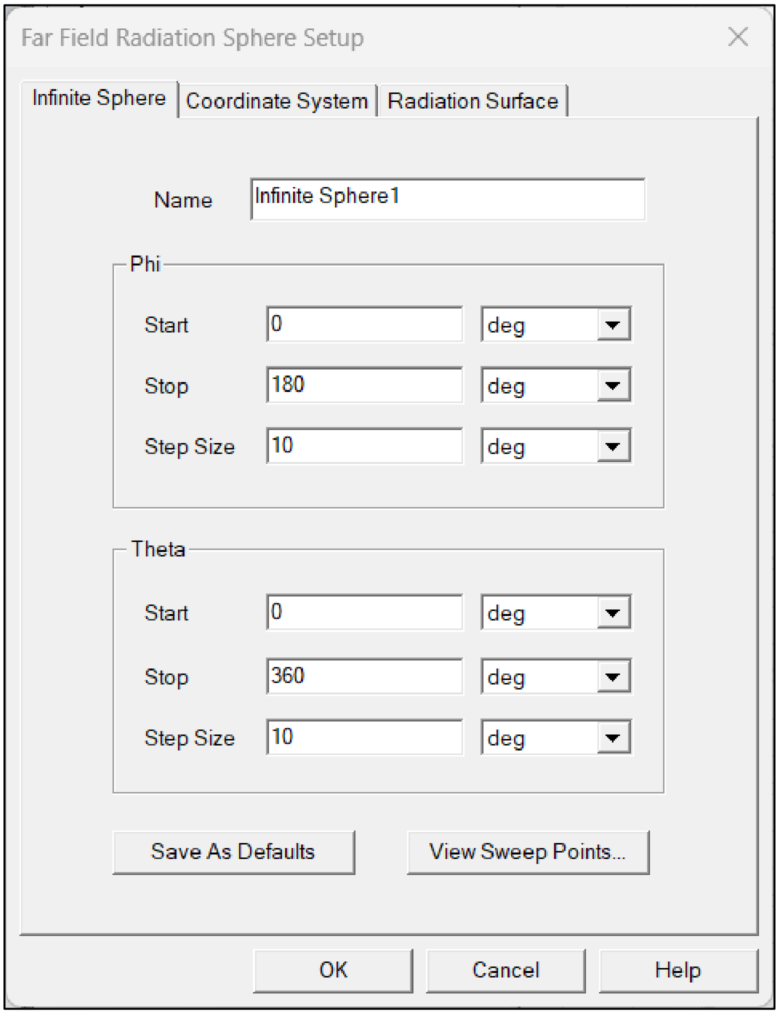


Figure 21: Radiation Infinite Sphere Settings

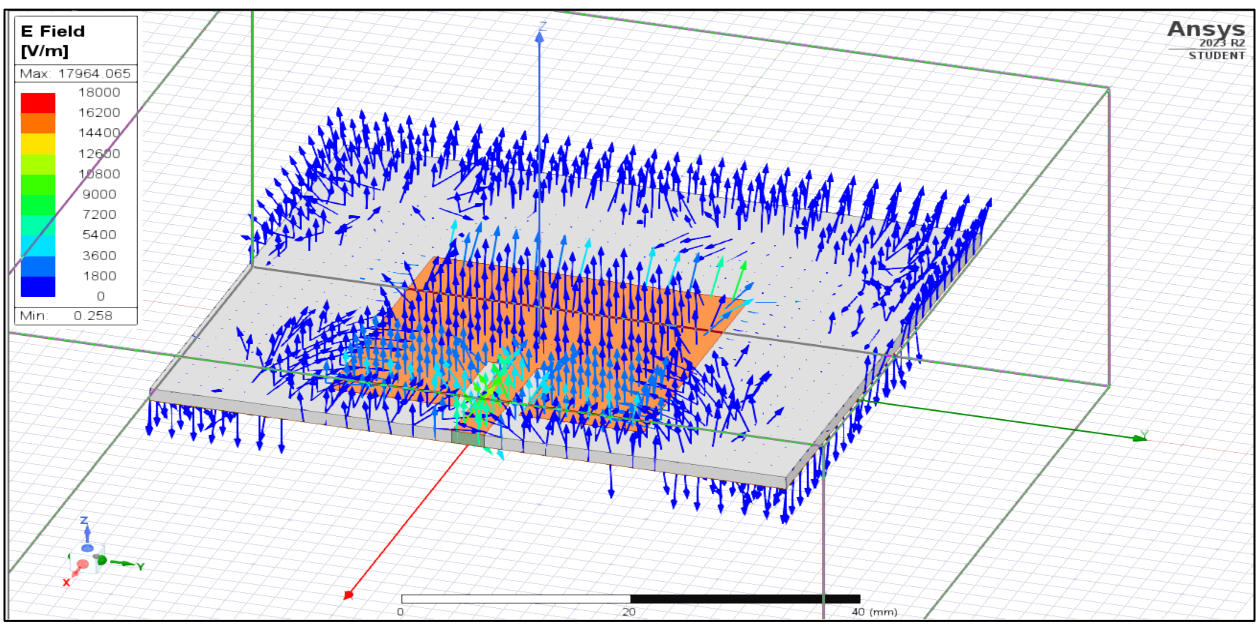


Figure 22: E-field Vector on the Patch Antenna as 2.4GHz

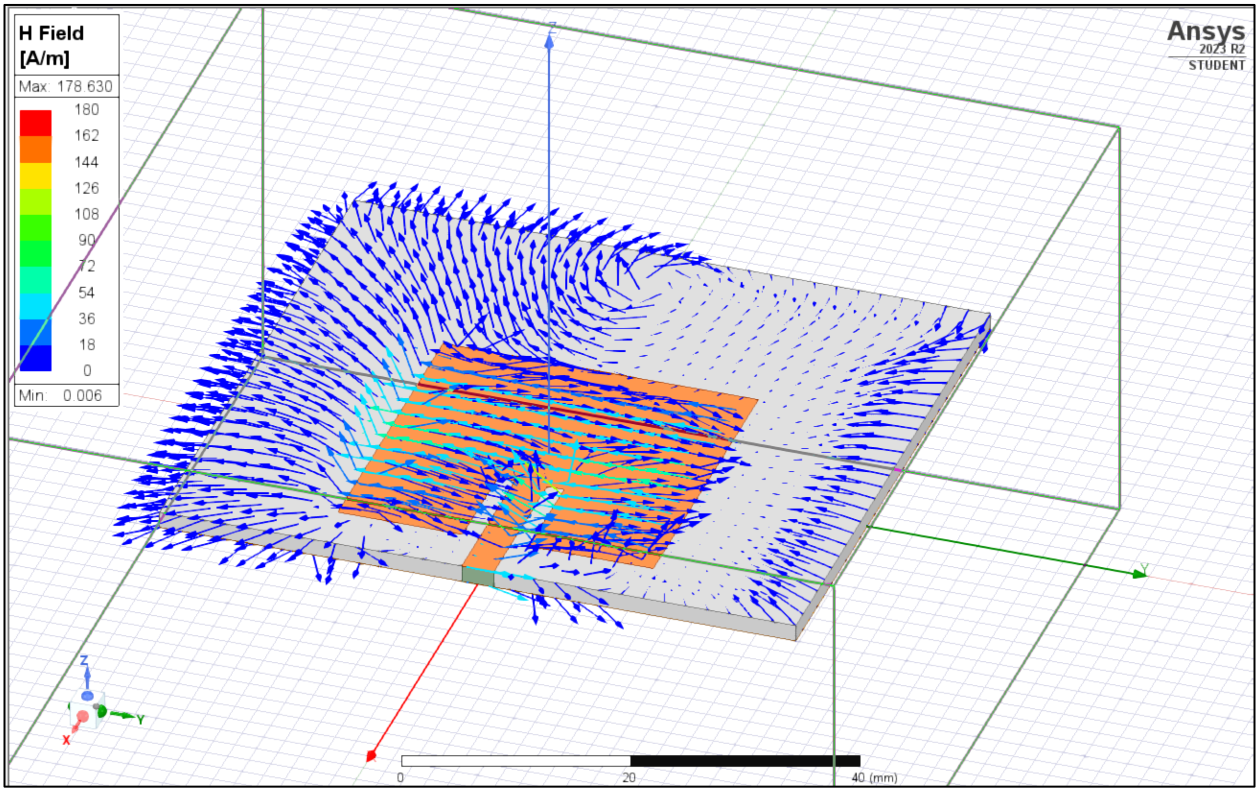


Figure 23: H-field vectors of the Patch Antenna at 2.4 GHz

PCB Gerber Files

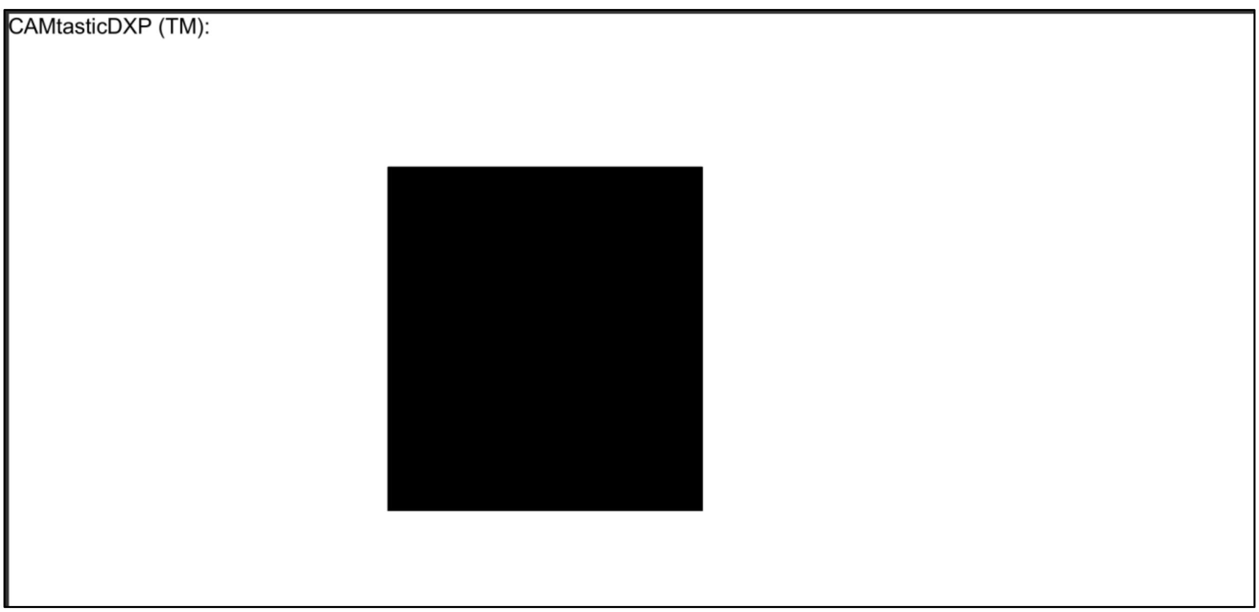


Figure 24: PCB Gerber File of the Patch Antenna Ground Plane

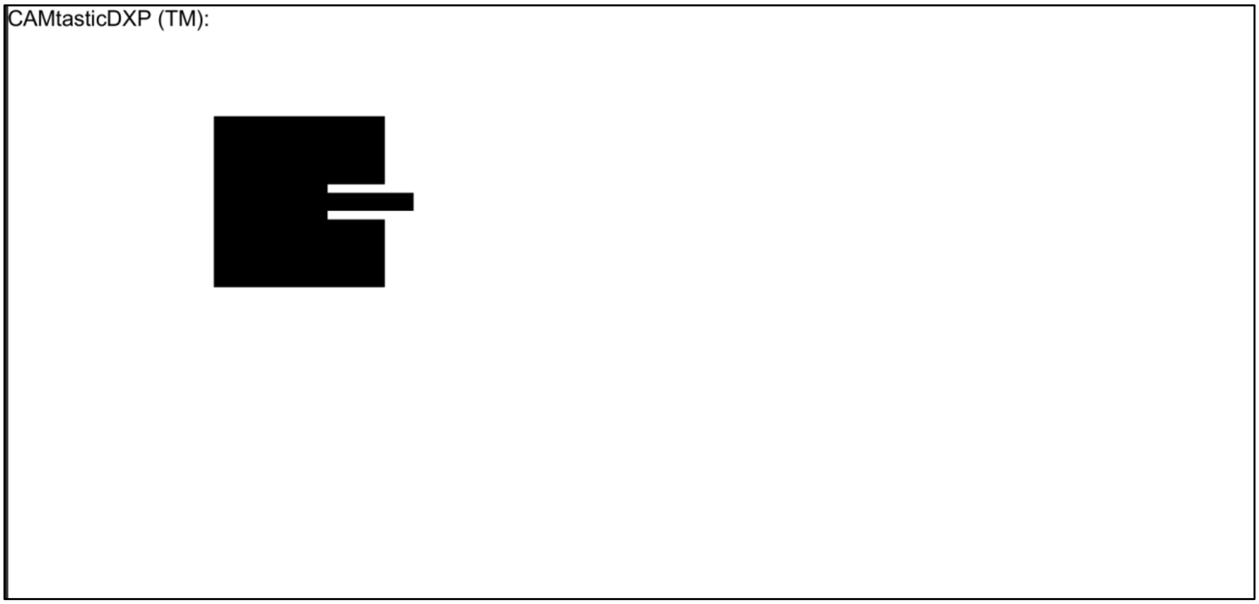


Figure 25: PCB Gerber File of the Top Square Patch

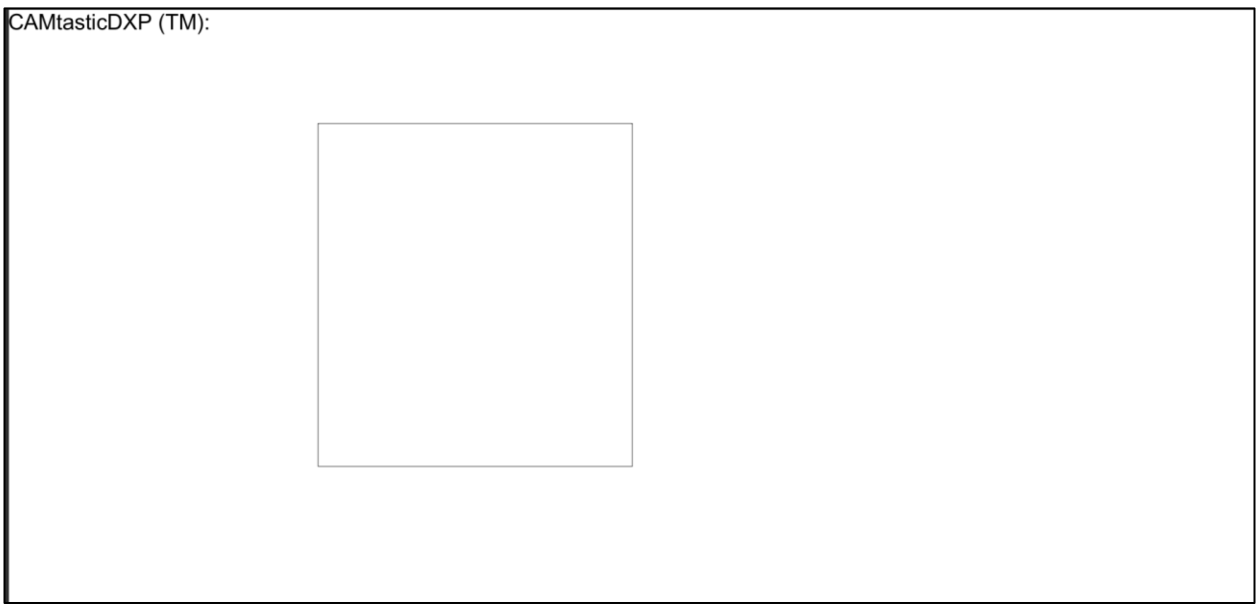


Figure 26: PCB Gerber File of the PCB Outline. We used this to create an outline of the PCB and cut out a piece out of the larger clad board.

Double-Sided Copper Clad Board:

A double-sided copper clad board is a type of printed circuit board (PCB) that features a layer of copper on both the top and bottom surfaces. These copper layers are bonded to an insulating substrate, typically made from materials such as fiberglass-reinforced epoxy resin (FR-4). The dual copper cladding allows for more complex and denser circuit designs, providing ample space for routing electrical connections on both sides of the board. This configuration is commonly used in electronic devices where more intricate and high-performance circuitry is required.

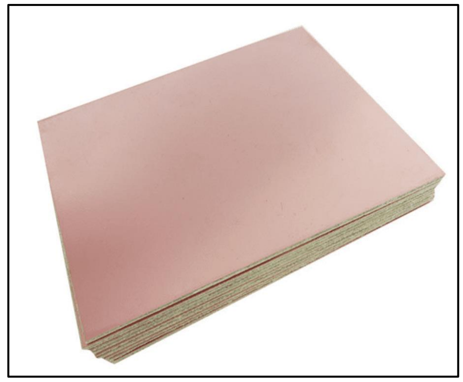


Figure 27: A double sided copper clad board was used to create the square-patch antenna

Ferric Chloride:

Ferric chloride (FeCl3) is a widely used etching solution for the production of printed circuit boards (PCBs). When applied to a copper clad board, ferric chloride selectively dissolves the exposed copper, removing unwanted areas and leaving behind the desired copper traces that form the circuit. This process typically involves submerging the PCB in a ferric chloride bath or applying the solution directly to the board. It is favored for its effectiveness and relatively low cost, though it must be handled with care due to its corrosive nature and potential environmental hazards. Proper safety measures, such as gloves and eye protection, are essential when using ferric chloride, and used solution must be disposed of in accordance with local regulations.

Figure 28: $FeCl_3$ was used during PCB etching. Any area of the clad board that is exposed , other than that covered by the PCB routes, will be washed away. This would leave behind a proper PCB circuit.

Clothing Iron:

Figure 29: The PCB Design was printed onto a butter paper, placed on a copper clad board and we iron the paper until we are sure that the design has been imprinted onto the copper clad board.

Using a clothing iron in PCB production is part of the toner transfer method, a popular DIY technique for creating printed circuit boards at home. In this process, a circuit design is printed onto glossy paper using a laser printer. The printed paper is then placed toner-side down on a clean copper clad board. A clothing iron, set to a high temperature without steam, is used to apply heat and pressure to the paper. This transfers the toner, which acts as an etch resist, onto the copper surface. Once the transfer is complete, the board is soaked in water to remove the paper, leaving the toner design on the copper. The board is then ready for etching, typically with ferric chloride, to create the final circuit. This method is favored for its simplicity and accessibility, making it ideal for hobbyists and small-scale prototyping.

Geometric Divider:

Figure 30: A geometric divider is used to create a proper outline on the PCB of the antenna.

A geometric divider is a precision instrument used primarily in drafting and geometry for measuring and transferring distances. It typically consists of two straight, pointed legs connected by a hinge, allowing the distance between the points to be adjusted. Dividers are useful for tasks such as marking equal intervals on a line, comparing lengths, and drawing circles or arcs with a specific radius. In technical drawing, they help ensure accuracy and consistency, making them an essential tool for engineers, architects, and artists. Modern versions may include features like a locking mechanism to maintain a set distance or a scale for direct measurements.

Hacksaw:

A hacksaw is a versatile hand tool that can be used to cut printed circuit boards (PCBs) to the desired size or shape. Featuring a thin, fine-toothed blade mounted in a metal frame, a hacksaw is ideal for precise cutting through the fiberglass or epoxy resin substrate of a PCB without causing excessive damage to the copper traces. To use a hacksaw effectively, secure the PCB firmly to prevent movement, and use smooth, steady strokes to achieve a clean cut. Proper safety measures, such as wearing eye protection, should be taken to avoid injury from fiberglass dust or sharp edges. This method is particularly useful for hobbyists and small-scale PCB fabrication where specialized cutting tools may not be available.

Figure 31: We used a hacksaw to cut through the copper clad board and get a small piece as required for our hardware.

Sanding Paper/ Silicon Carbide Sheet:

Sanding paper is a useful tool in the preparation and finishing of printed circuit boards (PCBs). It can be used for a variety of purposes in PCB fabrication, such as smoothing the edges after cutting, removing oxidation from the copper surface, and cleaning the board to ensure good adhesion of toner or resist materials. Fine-grit sandpaper (such as 400 to 600 grit) is typically recommended for these tasks to avoid scratching or damaging the copper layer. Proper sanding helps improve the quality of the etching process and the overall appearance of the PCB, contributing to better electrical performance and reliability. Always ensure that the board is thoroughly cleaned of any sanding debris before proceeding to the next steps in the PCB fabrication process.

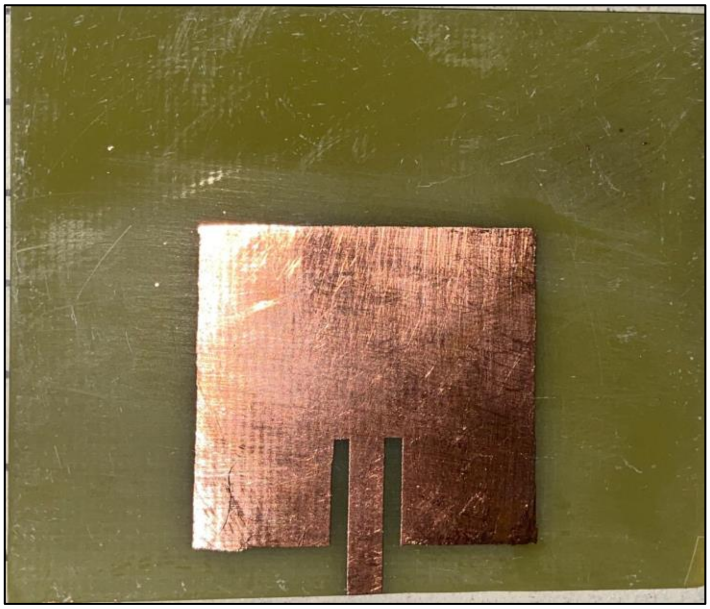
Figure 32: Silicon Carbide Sheet was used by us to give proper finishing to our PCB after cutting it using a hacksaw

Cleaning Agents:

A cleaning agent is essential in the PCB fabrication process to remove contaminants such as flux residues, oils, dust, and oxidation from the board. Commonly used cleaning agents include isopropyl alcohol (IPA), which is effective at dissolving flux residues and evaporates quickly without leaving a residue. Other specialized PCB cleaners and degreasers are also available, designed to clean more stubborn contaminants while being safe for electronic components. When using any cleaning agent, it is important to apply it with lint-free cloths or brushes to avoid introducing new particles onto the board. Proper cleaning ensures reliable electrical connections, improves solderability, and enhances the overall performance and longevity of the PCB.

Figure 33: Cleaning agent was used to remove any $FeCl_3$ from the PCB so that there is no further corrosion.

Project Hardware



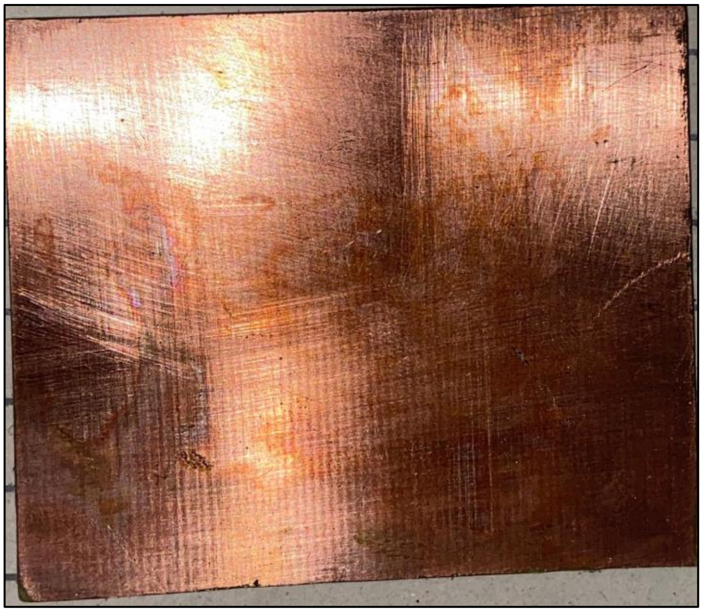


Figure 34: The square patch (left) and the ground plane of the patch antenna (right) .

Conclusion

In conclusion, the design and implementation of a microstrip inset feedline square patch antenna for 2.4 GHz involve meticulous planning and execution. Utilizing FR4 as the substrate and maintaining a dielectric height of 1.6 mm, the design aims to achieve a minimum bandwidth of 50 MHz and a gain of 4-5 dBi. HFSS simulations validate the design, followed by PCB fabrication using etching techniques. The process includes precise calculations for the antenna dimensions, effective permittivity, and fringing effects. The final steps involve transferring the design to PCB layout software, fabricating the PCB, mounting components, and testing the antenna's performance. Comprehensive documentation throughout the project ensures accuracy and facilitates any necessary optimizations, leading to a successfully designed and fabricated antenna.