Microstrip Patch Antenna design for Telemedicine Application

B.E. Dissertation

Submitted in partial fulfillment of the requirement of

University of Mumbai

For the Degree of

Bachelor of Engineering
(Electronics and Telecommunication Engineering)

by

Pathan Rukhsar Imtiyajkhan(15DET114)
Shaikh Mariya Arif(15DET123)
Sayyed Asim Shafique Ahmed(15DET117)
Aaga Faaz Aezaaz(14DET067)

Under the guidance of

Prof. Khadse Rahul

Department of Electronics and Telecommunication Engineering
Anjuman-I-Islam's Kalsekar Technical Campus,
Sector 16, New Panvel , Navi Mumbai
(Affiliated to University of Mumbai)
April 2018

Anjuman-I-Islam's
Kalsekar Technical Campus
(Affiliated to the University of Mumbai)
Plot 2 and 3,Sector 16, Khandagaon, Near Thana Naka, New Panvel, Navi Mumbai 410206.

Certificate

This is to certify that, the dissertation titled

"Microstrip Patch Antenna design for Telemedicine Application "

is a bonafide work done by

Pathan Rukhsar Imtiyajkhan(15DET114)
Shaikh Mariya Arif(15DET123)
Sayyed Asim Shafique Ahmed(15DET117)
Aaga Faaz Aezaaz(14DET067)

and is submitted in the partial fulfillment of the requirement for the degree of

Bachelor of Engineering
in
Electronics and Telecomunication Engineering
to the
University of Mumbai.

Guide Project Coordinator

Head of Department Principal

Certificate of Approval by Examiners

This is to certify that the dissertation entitled **"Microstrip Patch Antenna design for Telemedicine Application"** is a bonafide work done by **Pathan Rukhsar,Shaikh Mariya,Sayyed Asim,Aaga Faaz** under the guidance of **Prof. Khadse Rahul**. This dissertation has been approved for the award of **Bachelor's Degree in Electronics and Telecommunication Engineering**, University of Mumbai.

Examiners:

Name: Name:

Acknowledgments

We would like to record our appreciation to all people that involve in the writing this report. First of all, our appreciation goes to Electronics and Telecommunication Department for all the guidance especially our teacher, **Asst Prof. Afzal Shaikh** for guiding and supporting us until we complete this report successfully. In addition, a lots of love and thanks to our parents who always supported and encouraged us with their loves and hopes. Besides, we are also very grateful to all of our friends especially our classmates for all of the information and help. Certainly, thanks to our entire respondent for their co-operation in answering our questionnaire.

Abstract

The study of microstrip patch antennas has made great progress in recent years. Compared with conventional antennas, microstrip patch antennas have more advantages and better prospects. They are lighter in weight, low volume, low cost, low profile, smaller in dimension and ease of fabrication and conformity. Moreover, the microstrip patch antennas can provide dual and circular polarizations, dual-frequency operation, frequency agility, broad band-width, feedline flexibility, beam scanning omnidirectional patterning. Telemedicine is the use of telecommunications technology as a medium to provide live, interactive audiovisual medical services for sites that are at a distance from the provider. Telemedicine and related healthcare technologies aim to provide efficient healthcare remotely. The objective of this paper is to provide a better solution for telemedicine application. Various wireless technologies are used, but microstrip patch antenna is the best solution. In this paper microstrip patch antenna is designed. The design consists of microstrip patch antenna. This antenna is simulated on High frequency structure simulation (HFSS).

List of Figures

Contents

Chapter 1

Introduction

1.1 Introduction

Telemedicine means literally medicine at a distance. New technologies in sensing, medical imaging and wireless data communications are allowing telemedicine to provide healthcare at a distance with much lower cost than in the past, enabling the development of new widespread remote medicine initiatives.Researches categorize the telemedicine history into three eras.The first era can be named as telecommunications era of the 1970s. Applications in this era were dependent on broadcast and television technologies where telemedicine application was not integrated with any other clinical data. The second era of telemedicine, dedicated era, started during the late 1980s as a result of digitalization in telecommunications and it grew during 1990s The transmission of data was supported by various communication mediums ranging from telephone lines to Integrated Service Digital Network (ISDN) lines. Dedicated era has turned into an Internet era where more complex networks are supporting the telemedicine. The third era of telemedicine is supported by the technology that is cheaper and accessible to an increasing user population. The enhanced speed and quality offered by Internet or3G mobile telephony is providing new opportunities in telemedicine. Certain recent research projects include the use of satellite-based Telemedicine solutions. Satellite-based telemedicine services are used to solve teleconsultation, tele-education, home care, second opinion and other medical problems.There are many challenges in wireless monitoring of patients, including the coverage, reliability and quality of monitoring. One of the most difficult challenges in patient monitoring using wireless networks, especially for emergency messages, is the reliability of message delivery.Many hospitals and nursing homes are deploying infrastructure-oriented wireless networks, such as wireless LANs, satellites, and cellular and GSM in telemedicine systems range from simple heart rate, blood pressure, body temperature to blood glucose levels and ECG wave forms.

1.2 General structure of Microstrip Patch Antenna

A microstrip antenna generally consists of a dielectric substrate sandwiched between a radiating patch on the top and a ground plane on the other side. The patch is generally made of conducting material such as copper or gold and can take any possible shape. The radiating patch and the feed lines are usually photo etched on the dielectric substrate.

1.2.1 Types of Patch Antenna

These antennas can be made of various designs. The basic shapes like circular, rectangular, square, semi-circular etc. form a major section of the research in this field. They can be modified according to the use by adding slots, monopoles, arrays etc. Shown below are examples of some basic types of patch antennas.

Figure 1.1: Common Shapes of Microstrip Patch Elements

1.2.2 Why PAtch Antenna

x These antennas are lightweight, small in size and low profile. x Both liner and circular polarizations can be obtained using them. x Can be made compact. x Ease of mass production using the printed circuits makes them a cheaper option to use. x Major advantage is that they can work in multiband of frequencies.

1.3 Feeding Techniques

Microstrip patch antennas can be fed by a variety of methods. These methods can be classified into two categories- contacting and non-contacting. In the contacting method, the RF power is fed directly to the radiating patch using a connecting element such as a microstrip line. In the non-contacting scheme, electromagnetic field co WHY upling is done to transfer power between the microstrip line and the radiating patch. The four most popular feed techniques used are the microstrip line, coaxial probe (both contacting schemes), aperture coupling and proximity coupling (both non-contacting schemes).

1.3.1 Microstrip Line Feed

In this type of feed technique, a conducting strip is connected directly to the edge of the microstrip patch as shown in figure. The conducting strip is smaller in width as compared to the patch. This kind of feed arrangement has the advantage that the feed can be etched on the same substrate to provide a planar structure.

An inset cut can be incorporated into the patch in order to obtain good impedance matching without the need for any additional matching element. This is achieved by properly controlling the inset position. Hence this is an easy feeding technique, since it provides ease of fabrication and simplicity in modeling as well as impedance matching. However as the thickness of the dielectric substrate increases, surface waves and spurious feed radiation also increases, which hampers the bandwidth of the antenna. This type of feeding technique results in undesirable cross polarization effects.

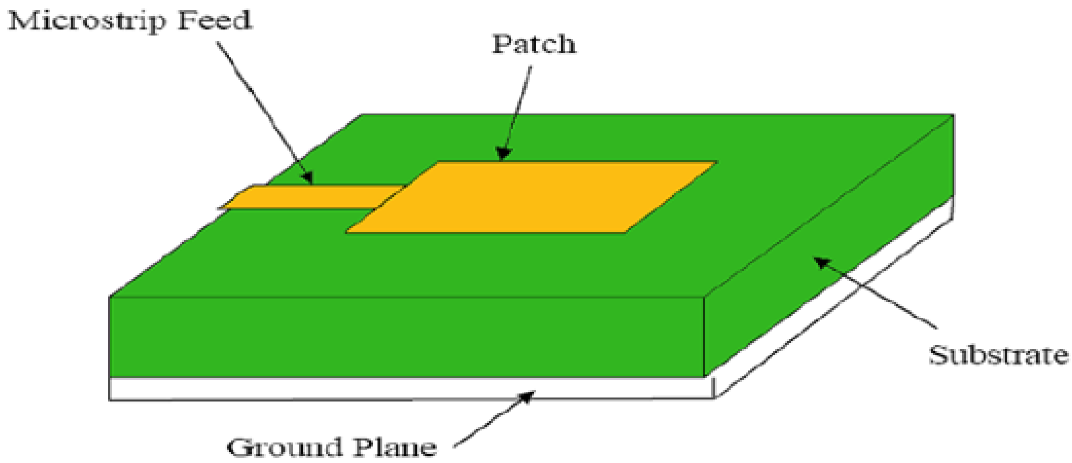

Figure 1.2: Microstrip Line Feed

1.3.2 Coaxial Feed

The Coaxial feed or probe feed is one of the most common techniques used for feeding microstrip patch antennas. As seen from figure, the inner conductor of the coaxial connector extends through the dielectric and is soldered to the radiating patch, while the outer conductor is connected to the ground plane.

The main advantage of this type of feeding scheme is that the feed can be placed at any desired position inside the patch in order to obtain impedance matching. This feed method is easy to fabricate and has low spurious radiation effects. However, its major disadvantage is that it provides narrow bandwidth and is difficult to model since a hole has to be drilled into the substrate. Also, for thicker substrates, the increased probe length makes the input impedance more inductive, leading to matching problems.

By using a thick dielectric substrate to improve the bandwidth, the microstrip line feed and the coaxial feed suffer from numerous disadvantages such as spurious feed radiation and matching problem. The non-contacting feed techniques which have been discussed below, solve these problems.

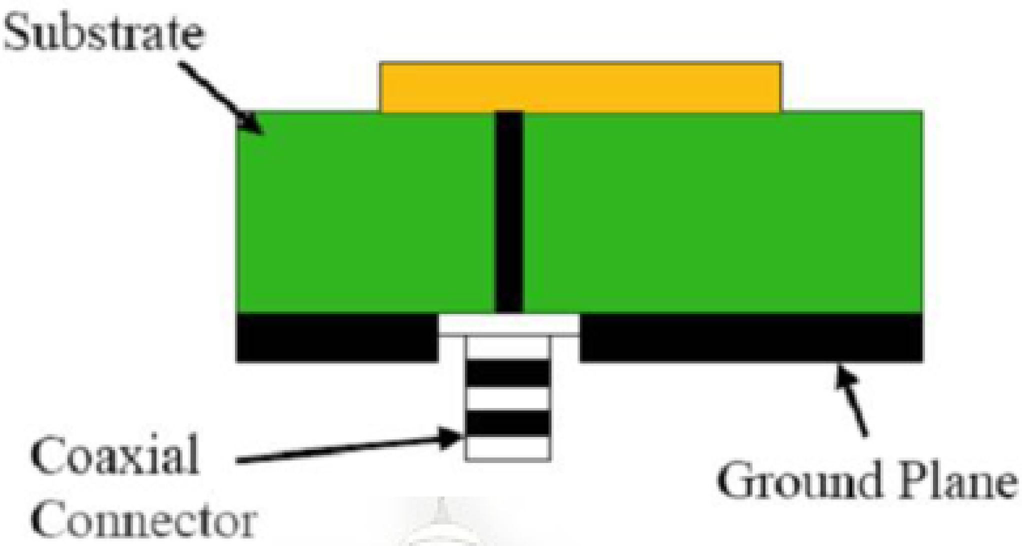

Figure 1.3: Coaxial Feed

1.3.3 Aperture Coupled Field

In aperture coupling as shown in figure, the radiating microstrip patch element is etched on the top of the antenna substrate, and the microstrip feed line is etched on the bottom of the feed substrate in order to obtain aperture coupling. The thickness and dielectric constants of these two substrates may thus be chosen independently to optimize the distinct electrical functions of radiation and circuitry. The coupling aperture is usually centered under the patch, leading to lower cross-polarization due to symmetry of the configuration. The amount of coupling from the feed line to the patch is determined by the shape, size and location of the aperture. Since the ground plane separates the patch and the feed line, spurious radiation is minimized.

Generally, a high dielectric material is used for bottom substrate and a thick, low dielectric constant material is used for the top substrate to optimize radiation from the patch. This type of feeding technique can give very high bandwidth of about 21

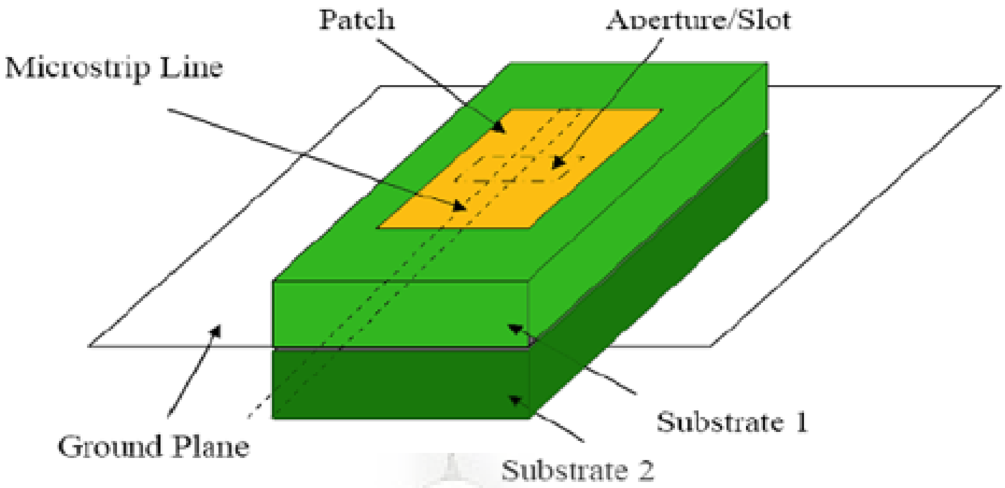

Figure 1.4: Aperture Coupled Feed

1.3.4 Proximity Coupled Feed

This type of feed technique is also called as the electromagnetic coupling scheme. As shown in figure, two dielectric substrates are used such that the feed line is between the two substrates and the radiating patch is on top of the upper substrate. The main advantage of this feed technique is that it eliminates spurious feed radiation and provides very high bandwidth of about 13

The major disadvantage of this feed scheme is that it is difficult to fabricate because of the two dielectric layers that need proper alignment. Also, there is an increase in the overall thickness of the antenna.

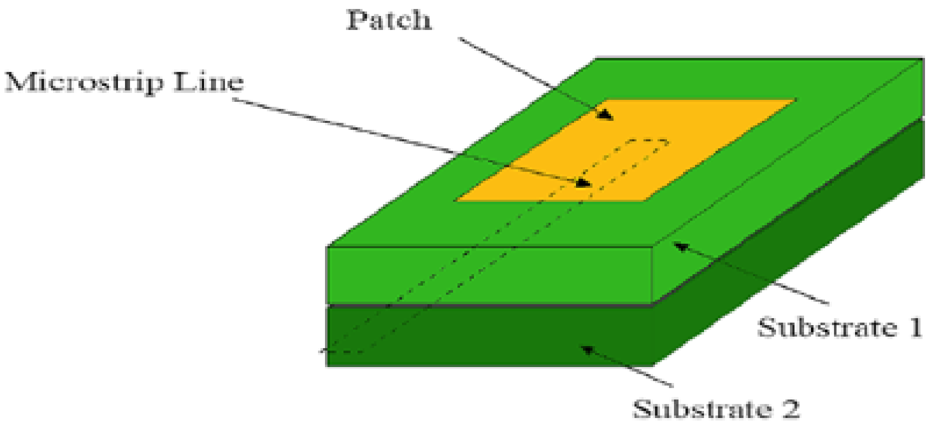

Figure 1.5: Proximity Coupled Feed

Chapter 2

Hardware Design

2.1 Printed Circuit Board

PCB is an acronym for Printed Circuit Board it is a board that connects various points of the circuit together and allows signals and power to be routed between different components. A bare PCB is also called as copper clad. Earlier circuits were made using point to point wiring which made them time consuming to manufacture, very difficult to debug and prone to failures.As technology improved components became smaller also there was need for large scale production as well as cost reduction all these were major factors that lead to the concept of PCB being born. If you are interested in knowing the History of PCB do check out this link. PCB Board is one of the most basic part of any electronic circuit. Just remember that any electronic device/product/part in a product is always mounted on a PCB.

Figure 2.1: Printed Circuit Board

A PCB board is etched from bare PCB also called as Copper clad which are basically copper sheets laminated onto a non-conductive substrate. A bare Copper clad consists of two parts

Substrate: The base material also called Substrate or laminate is a non-conductive layer provides strength and flexibility to the PCB. Various materials are used as substrate later in this article we will discuss the material used for substrate. This base material can be flexible as well.

Copper Foil: The copper sheet or copper foil is the conductive part of the PCB and it is this part that allows current to flow through the circuit. Other Conductive material like stainless steel,beryllium copper or nickel can also be used but copper is used extensively because it is easily available and comparatively is low costing.

A copper clad can be classified based upon various parameters/factors

Base Material Used: The base material provides insulation, mechanical strength, stability and rigidity to the copper clad. Two of the most common type of base material used are:

1.Paper Phenolic:

These PCB material are known as FR1 and FR2 PCB. In this type of PCB Paper (mixture of wood fiber and phenolic polymers) is used as a reinforcement. Paper Phenolic PCB is mostly brown in colour. They are low cost but have tendency to absorb moisture and they crack/shears easily. They are used mostly by students, hobbyist and in products where cost is a big factor.

2.Glass Epoxy PCB:

These PCB material are known as FR4 PCB in these type of PCB fiber glass is used as base material. Glass Epoxy PCB's are generally greenish in colour. Glass epoxy PCB have near zero water absorption, chemical resistant, flame resistant and have higher electrical insulating qualities. They are costlier than paper phenolic PCB.

Copper foil Thickness: The conductive layer of a PCB is made of copper. The thickness of the copper is usually expressed in ounces per square foot. Well it is odd isn't it? Copper thickness is being measured in ounce. In PCB terminology 1 ounce/square ft is equivalent to 35 microns (1 micron is one millionth of a meter) similarly 2 ounce/square ft is equivalent to 70 microns. As the thickness of the copper foil increases its resistivity decreases meaning lesser loss and lesser heating of tracks. But the price of the copper clad increases as the thickness increases. So according to your application you decide the copper foil thickness. Most of the commonly used PCB's have copper thickness of 18 micron. And if you want to reduce cost further you can get PCB's up-to 12 micron thickness. So depending on your application you can select the foil thickness of your copper clad.

Number of Layers: PCB's can also be classified based on the number of copper layers on one PCB.

1.Single Sided PCB:

In single sided PCB there is only one layer of copper foil and one layer of substrate. Normally in a single sided PCB there is top side and bottom side. Top side is where through hole components are mounted and the bottom side is where soldering of components leads are done. SMD components are also soldered on bottom side.

Figure 2.2: Single Layer PCB

Single sided PCB's are low cost and very easy to manufacture, test and debug. These are so easy to manufacture that you can manufacture them at your home with some basic raw material.

2.Double Sided PCB

Double sided PCB's are a bit difficult to manufacture. In this type there is only one base material but there are two copper layers one on both sides of the base material. So basically both sides of Double sided PCB's are conductive and electronic components can be placed as well as soldered on both sides of the PCB. There are tracks on both sides of the PCB and are connected together using PTH technology.

2.1.1 What exactly is PTH PCB?

PTH is an abbreviation for "Plated through Holes". In double/multi layer PCB's for connecting tracks electrically on different layers of the board drill holes with conductive material are made. These conductive holes are also called as Vias. These PTH Connections either form simple electrical connection between both sides of the PCB (Via Holes),

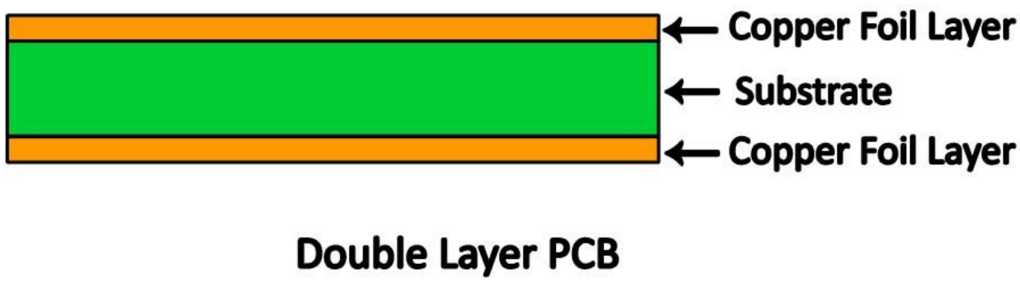

Figure 2.3: Double Layer PCB

or electrical connectivity and also provide good mechanical support for leaded components. This makes the double sided PTH PCB a much more physically robust item. Though you can make a double sided PCB without PTH as well. These type of PCB is called as Double sided non-PTH PCB. These types of PCB's are rarely used. In this case the electrical connection between the two layers is done by manually i.e. placing a link (normally a resistor leg is used) in a drill hole and then soldering on both sides of the board.

3.Multi-Layers PCB

Multi layer PCB's are the upper version of double sided PCB. In this type more than two copper layers are there on a single PCB. Multi-layer PCB provides designers to manufacture very complex and compact circuits having high component density.

A multi layer PCB is basically multiple double sided PCB which are bonded together with a special glue called as pre-leg. In multi layer PCB's separate layers are used for power and ground and helps in decreasing the electromagnetic interference and lowering the EMI. PCB boards can also be classified into flexible and flex rigid PCB .Will try to cover this type some other time. Also now a days a Metal Clad PCB are used specially in Lighting industry where high wattage of LED's are used. This type of PCB's have Aluminium backing i.e. there are three layers in this type of PCB One is the copper layer, a insulating layer and a Aluminium/Metal layer which provides very high heat

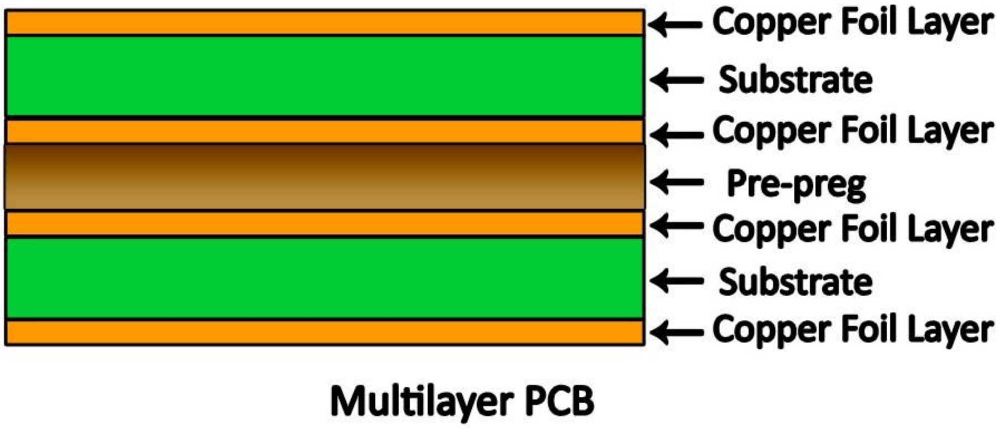

Figure 2.4: MultiLayer PCB

dissipation as compared to FR4 PCB board.

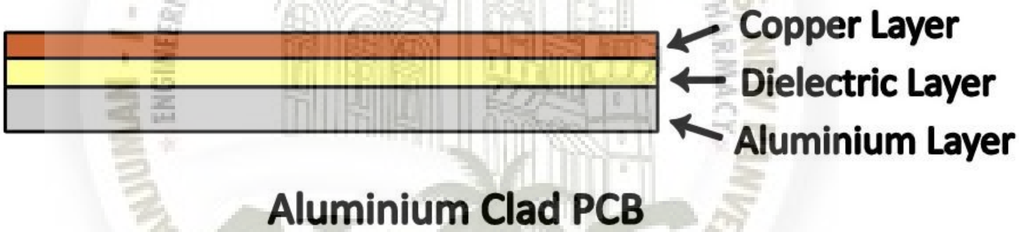

Figure 2.5: Aluminium Clad PCB

Features and details Layers : Double Sided; Surface Finishing : Copper Clad; Material : FR4 Glass Fiber; Product information ModelNumber- CCL10X15DSFR4 Number of Memory Sticks- 2 Item model number- CCL10X15DSFR4

2.2 Etching

Most PCBs are made from a material called FR4, which is a glass-reinforced epoxy composite (basically a sheet of fiberglass) with copper traces on one or both sides for carrying signals. While it may look like the copper traces are printed onto a blank sheet of fiberglass, the fact of the matter is that circuit boards usually start as a fully copper-plated sheet. You can buy this stuff online for super cheap, just do an Amazon search for copper clad board. In order to isolate the traces, all of the excess copper needs to be removed. This can be done with a mill or router like the Shapeoko, but traditionally its been done chemically. The problem is, your chemical etchant doesnt know where your traces are supposed to be, so you need to mask them off and keep them away from the etchant.

Masking the Board: The easiest method for getting your layout onto a piece of copper-clad is a method called toner transfer, where you print your design onto transfer paper using a laser printer and then transfer that toner using a press or an iron. Transfer paper is basically a plastic substrate coated in a powdery blue finish. You can order just a few sheets from sellers on eBay and Amazon. If you dont have the time and money to order it online, you can substitute a glossy magazine page, and Ive gotten some pretty decent results that way. When youre printing your design, be sure that youve mirrored it so that after you transfer it face down, the design is oriented correctly. Also, print on a normal sheet of paper first to check your scaling and position. Prepare your piece of copper-clad board by wiping it down with some alcohol. This will help remove oils and gunk that might keep the toner from sticking. If your board has been sitting in a humid environment for a while, it might need a quick shine with a scouring pad. Now cut out your design and place it face down on your freshly cleaned board. You can stick it in place with a few piece of masking tape. If you have a desktop laminator, this next step is really easy. Just set your laminator to its hottest and slowest setting (usually a preset for 10mil pouches) and run your board with the transfer taped to it through the laminator four or five times. If you dont have access to a laminator, you can use a clothes iron set on high with no steam. Apply even pressure to the board for maybe five minutes and dont allow the transfer paper to move or slip against the board. It helps to put a piece of paper between the iron and the transfer sheet. Once your board has cooled off, peel away the transfer paper to reveal your resist! There way be spots that didnt transfer cleanly; those can be filled in with a permanent marker.

Etching the board: There are a few different etchant solutions you can use to remove copper from a piece of copper-clad board. The most common (old-school) solution is ferric chloride. Ferric chloride is toxic, highly corrosive and acidic, but its also inexpensive and easy to get your hands on. I prefer to use it because, well I know how to judge how fast its going to etch, and Im familiar with it. Another popular solution is cupric chloride, which can also be toxic, but a lot of people prefer it because its etching strength actually increases as more copper is dissolved into it (to a degree), and it can be regenerated by aeration with an aquarium pump. A solution thats really easy to get your hands on and is rumored to work pretty well is a mixture of muriatic acid (dilute hydrochloric acid from a pool and spa supplier) and hydrogen peroxide. If youre worried about the toxicity of these solutions and want a slightly safer option you can try using a mixture of vinegar, table salt and hydrogen peroxide, although Ive tried this in the past (out of curiosity) and its really slow and uneven. You dont need to be scared of toxic solutions, just be safe. Wear gloves, wear goggles, dont put the solution into containers you plan to ever use for food remember high school chemistry lab. That being said, you can get hurt if you do something stupid so be careful. And, by the way, never pour used etchant solutions down the drain no matter which etchant solution you choose. The concentration of dissolved copper is bad for the environment and rough on water processing equipment. Throw on your personal protective equipment and dunk that masked board in the etch bath. This is gonna take a few minutes. One thing thats important is to keep the board moving in the solution. As the etchant reacts with the copper, it leaves byproducts that inhibit the etching process, and the best way to keep that stuff from settling is to keep the solution moving around. I stir it with a plastic knife. You can also build etching tanks just for this process which either rock your etch bath back and fourth or bubble air up through the solution to agitate it. If youre only making a few boards here and there, I would argue that stirring by hand is better because youre going to be pulling the board out and checking on it pretty regularly anyway. You can tell when the board is done etching because (surprise) the copper will be gone! As soon as you cant see any copper, pull that board out of the etchant because it will tunnel its way in sideways under your etch resist and thin out the traces. Rinse the board with some water to get rid of all of the excess etchant, and then you can take off those awful, sweaty gloves.

Removing the resist: This step only deserves its own section because its so beautiful and satisfying. Take a paper towel with a little acetone on it (it doesnt have to be concentrated, nail polish remover works fine) and wipe the etch resist away; it will come rightoff and reveal all those sweet, shiny traces!

Finishing Touches: Your board is pretty much done, now all you need to do is populate it. If your design requires through-hole parts, youll need to drill all of the holes.

I used a tiny drill bit on my rotary tool for which I happen to have a fancy drill press. You can also just do this with a hand drill if youre super careful. Then add your parts and solder everything in place!

2.3 SubMiniature version A Connector

SMA (SubMiniature version A) connectors are semi-precision coaxial RF connectors developed in the 1960s as a minimal connector interface for coaxial cable with a screw-type coupling mechanism. The connector has a 50 impedance. SMA is designed for use from DC to 18 GHz, but is most commonly used for hand-held radio and mobile telephone antennas, and more recently with WiFi antenna systems and USB software-defined radio dongles[citation needed]. It is also commonly used in radio astronomy, particularly at higher frequencies (5GHz+). SMA connectors can be visually confused with the standard household 75-ohm type F coax connector (diameters: Male 716 inch (11 mm) circular or hex; female 38 in (9.5 mm) external threads), as there is only about a 2 mm difference overall in the specifications. Type F cannot be mated with SMA connectors without the use of an adapter. The SMA name is also used for a superficially similar optical fiber connector.

2.3.1 Connector design

The SMA connector employs a 1/4-inch diameter, 36-thread-per-inch threaded barrel. The male is equipped with a hex nut measuring 5/16 inch (0.3125 inch / 7.9 mm) across opposite flats, thus taking the same wrench as a 6 SAE hex nut. A standard-polarity SMA male connector has a center pin surrounded by barrel with inside threads, and the standard SMA female connector has a center sleeve surrounded by a barrel with outside threads. As with most other connectors, the gender assignment thus corresponds to the innermost electrical component. There are also reverse-polarity (”RP”) SMA connectors in which the pin and sleeve are swapped; so that the ”male” RP-SMA has a center sleeve surrounded by an inside-threaded barrel, and the ”female” RP-SMA has a center pin and an outside-threaded barrel. See below for a fuller description. The SMA connector uses a polytetrafluoroethylene (PTFE) dielectric which will contact along the mating plane. Variability in the construction and the mating of the connectors limit the repeatability

Figure 2.6: SubMiniature version A Connector

of the connector impedance. For that reason and that they are just rated for a limited number of connection cycles, an SMA connector is not usually a good choice for metrological applications.[2] SMA connectors are rated for up to 500 mating cycles,[3] but to achieve this it is necessary to properly torque the connector when making the connection. A 5/16-inch torque wrench is required for this, set to 35 inlbf (0.3 to 0.6 Nm) for brass, and 710 inlbf (0.8 to 1.1 Nm) for stainless steel connectors. Flats are sometimes also provided on the cable side of the connector assembly so that a second wrench can be used to prevent it from rotating and damaging the joint to the cable. It is also advisable to inspect and clean out loose debris from the internal surfaces with compressed air or a gas duster can before mating.[4][5] Using a threaded interface, SMA 50 Ohm connectors are semi-precision units that provide excellent electrical performance from DC to 26.5 GHz and outstanding mechanical durability. SMA connectors feature stainless steel or brass construction and - 36 threaded coupling, which offers high performance in a compact design. For phase array radar, test equipment, ILS landing systems and other instrumentation using phase matching techniques, SMA interconnects offer a precise and simple means of phase adjustment for microwave devices. Built in accordance with MIL-C-39012 and CECC 22110/111, SMA connectors can be mated with all connectors that meet these specification mating diameters regardless of manufacturer.

2.3.2 Features and Benefits

1.Light weight, compact and vibration proof design.

2.Low cost commercial grade (Brass SMA) available in nickel or gold plating .

3.Terminates to all standard flexible coaxial cables, low-loss (LMR) type cables and industry standard semi-rigid and conformable cables.

2.3.3 Applications

1.Base stations.

2.Antennas.

3.Telecommunications.

4.Instrumentation.

5.PC/LAN.

Chapter 3

Software Design

3.1 High Frequency Structure Simulator

HFSS is a commercial finite element method solver for electromagnetic structures from Ansys. The acronym stands for High Frequency Structure Simulator. It is one of several commercial tools used for antenna design, and the design of complex RF electronic circuit elements including filters, transmission lines, and packaging. It was originally developed by Professor Zoltan Cendes and his students at Carnegie Mellon University. Prof. Cendes and his brother Nicholas Cendes founded Ansoft and sold HFSS stand-alone under a 1989 marketing relationship with Hewlett-Packard, and bundled into Ansoft products. In 1997 Hewlett-Packard acquired Optimization Systems Associates Inc. (OSA), a company John Bandler founded in 1983. HP's acquisition was driven by the HP's need for an optimization capability for HFSS. After various business relationships over the period 1996-2006, HP (which became Agilent EEsof EDA division) and Ansoft went their separate ways: Agilent with the critically acclaimed FEM Element and Ansoft with their HFSS products, respectively. Ansoft was later acquired by Ansys. ANSYS HFSS software is the industry standard for simulating high-frequency electromagnetic fields. Its gold-standard accuracy, advanced solvers and high-performance computing technologies make it an essential tool for engineers tasked with executing accurate and rapid design in high-frequency and high-speed electronic devices and platforms. HFSS offers state-of the-art solver technologies based on finite element, integral equation, asymptotic and advanced hybrid methods to solve a wide range of microwave, RF and high-speed digital applications. HFSS delivers 3-D full-wave accuracy for components to enable RF and high-speed design. By leveraging advanced electromagnetic field simulators dynamically linked to powerful harmonic-balance and transient circuit simulation, HFSS breaks the cycle of repeated design iterations and lengthy physical prototyping. With HFSS, engineering teams consistently achieve best-in-class design in a broad range of applications including antennas, phased arrays, passive RF/mW components, high-speed interconnects, connectors, IC packaging and PCBs. Design sign-off accuracy is provided by HFSS through its groundbreaking and industry-leading adaptive meshing technology. Its powerful meshing and solver technologies enable you to design with confidence, knowing the results provided by HFSS can be relied on. Other tools simply give answers without any feedback regarding the accuracy of the solution, leading to uncertainty. When combined with ANSYS HPC technologies, like domain decomposition or distributed frequencies, HFSS can simulate at a speed and scale never before thought possible, further allowing you to more fully explore and optimize your devices performance. With HFSS you know your designs will deliver on their product promise.

The Six general Steps in an HFSS Simulation

In Brief There are six main steps to create and solve a proper HFSS simulation.

They are:

1. Create model/geometry
2. Assign boundaries
3. Assign excitations
4. Set up the solution
5. Solve
6. Post-process the results

In Detail Every HFSS simulation will involve, to some degree, all six of the above steps. While it is not necessary to follow these steps in exact order, it is good modeling practice to follow them in a consistent model-to-model manner.

Step One: The initial task in creating an HFSS model consists of the creation of the physical model that a user wishes to analyze. This model creation can be done within HFSS using the 3D modeller. The 3D modeller is fully parametric and will allow a user to create a structure that is variable with regard to geometric dimensions and material properties. A parametric structure, therefore, is very useful when final dimensions are not known or design is to be tuned. Alternatively, a user can import 3D structures from mechanical drawing packages, such as SolidWorks, Pro/E or AutoCAD. However, imported structures do not retain any history of how they were created, so they will not be parameterizable upon import. If parameterization of the structure is desired, a user will need to manually modify the imported geometry so that parameterization is possible.

Step Two: The assignment of boundaries generally is done next. Boundaries are applied to specifically created 2D (sheet) objects or specific surfaces of 3D objects. Boundaries have a direct impact on the solutions that HFSS provides; therefore, users are encouraged to closely review the section on Boundaries in this document.

Step Three: After the boundaries have been assigned, the excitations (or ports) should be applied. As with boundaries, the excitations have a direct impact on the quality of the results that HFSS will yield for a given model. Because of this, users are again encouraged to closely review the section on excitations in this document. While the proper creation and use of excitations is important to obtaining the most accurate HFSS results, there are several convenient rules of thumb that a user can follow. These rules are described in the excitations section.

Step Four: Once boundaries and excitations have been created, the next step is to create a solution setup. During this step, a user will select a solution frequency, the desired convergence criteria, the maximum number of adaptive steps to perform, a frequency band over which solutions are desired, and what particular solution and frequency sweep methodology to use.

Step Five: When the initial four steps have been completed by an HFSS user, the model is now ready to be analyzed. The time required for an analysis is highly dependent upon the model geometry, the solution frequency, and available computer resources. A solution can take from a few seconds, to the time needed to get a coffee, to an overnight run. It is often beneficial to use the remote solve capability of HFSS to send a particular simulation run to another computer that is local to the users site. This will free up the users PC so it can be used to perform other work.

Step Six: Once the solution has finished, a user can post-process the results. Post-processing of results can be as simple as examining the S-parameters of the device modelled or plotting the fields in and around the structure. Users can also examine the far fields created by an antenna. In essence, any field quantity or S,Y,Z parameter can be plotted in the post-processor. Additionally, if a parameterized model has been analyzed, families of curves can be created.

3.1.1 Application

1.Antennas.

2.Microwave transitions.

3.Waveguide components.

4.RF filters.

5.Three-dimensional discontinuities.

6.Passive circuit elements.

3.1.2 Features

1.Computes s-parameters and full-wave fields for arbitrarily-shaped 3D passive structures.

2.Powerful drawing capabilities to simplify design entry.

3.Field solving engine with accuracy-driven adaptive solutions.

4.Powerful post-processor for unprecedented insight into electrical performance.

5.Advanced materials.

6.Model Library-including spiral inductors.

7.Model half, quarter, or octet symmetry.

8.Calculate far-field patterns.

9.Wideband fast frequency sweep .

10.Create parameterized cross section models- 2D models.

3.1.3 Advantages

1.HFSS is an interactive simulation system whose basic mesh element is a tetrahedron. This allows you to solve any arbitrary 3D geometry, especially those with complex curves and shapes, in a fraction of the time it would take using other techniques.
2.Ansoft pioneered the use of the Finite Element Method (FEM) for EM simulation by developing/implementing technologies such as tangential vector finite elements, adaptive meshing.
3.HFSS has evolved over a period of years with input from many users and industries. In Industry, Ansoft HFSS is the tool of choice for High productivity research, development, and virtual prototyping.

3.1.4 Disadvantages

1.Narrow Bandwidth
2.Low Efficiency
3.Low Gain
4.Extraneous radiation from feeds and junctions
5.Poor end fire radiator except tapered slot antennas
6.Low power handling capacity
7.Surface wave excitation

Chapter 4

Project Operation

4.1 Working of Project

To analyse the radiation from a patch let us take an example of a rectangular patch considering it to be a two-dimensional planar structure. By definition a rectangular patch is defined by its length and width. The width of a patch is comparable to the wavelength and thickness of the substrate is kept very much smaller than wavelength. The fundamental TM10 mode can be attained by keeping the length slightly less than /2, where is wavelength in the dielectric medium (equal to 0 /, where is effective dielectric constant and 0 is the free space wavelength). The effective dielectric constant is kept slightly less than the dielectric constant of the substrate so that the fields not entirely get confined to the substrate but also fringe and spread in the air.

In TM10 mode, along the length, fields along the length are 180 degree out of phase with each other and cancel out. There is no variation is observed along the width of the patch . If the field along the width is resolved into components, the vertical components would cancel out and horizontal components add up resulting in a broadside pattern as shown in figure. Hence, the width edges are termed as radiating edges and length edges as non-radiating edges.

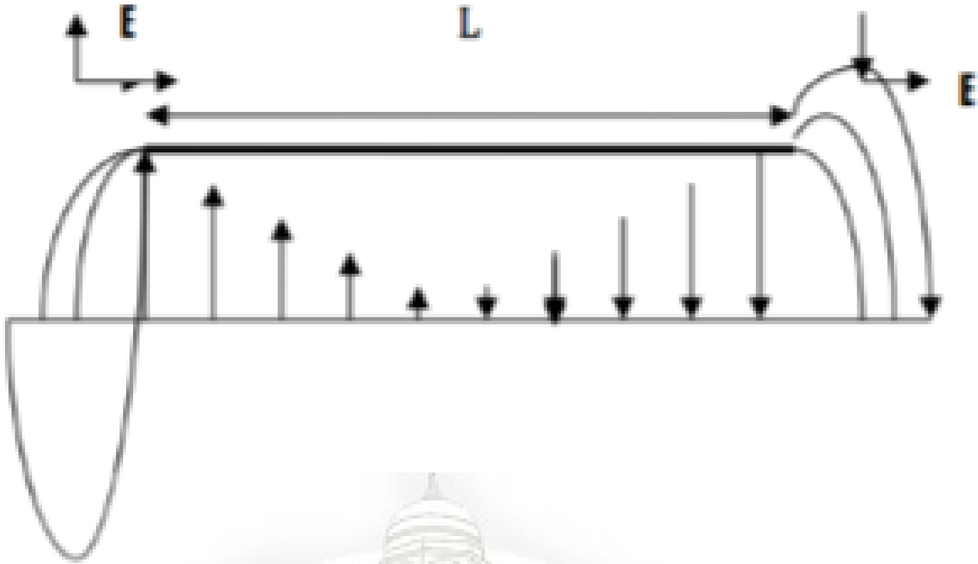

Figure 4.1: E Field

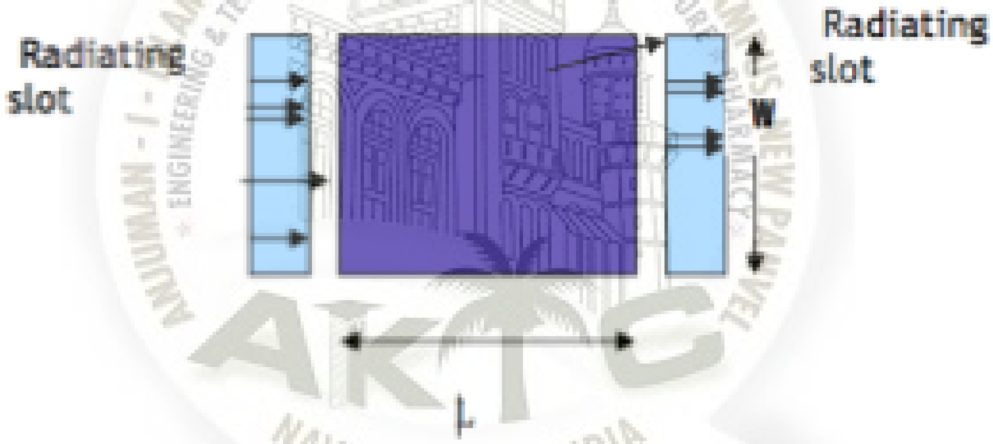

Figure 4.2: Variation of Feed Lines

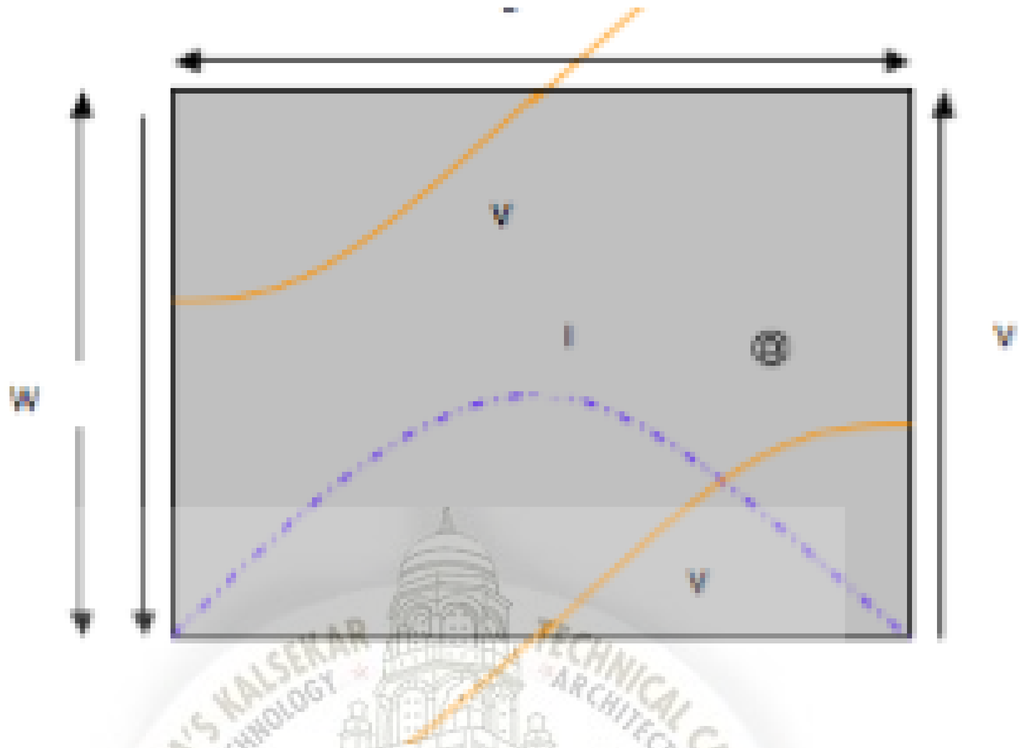

Figure 4.3: Field Pattern

Now a patch, which operates in TM10 mode, can be visualized and analyzed as a transmission line, as the field lines vary sinusoidally along the length while are uniform along the width. The fringing fields along the edges can be explained and modeled as equivalent capacitance and radiation resistance of the transmission line as shown in figure. Thus, one of the basic methods used in the analysis of patch is by analyzing it as a transmission line.

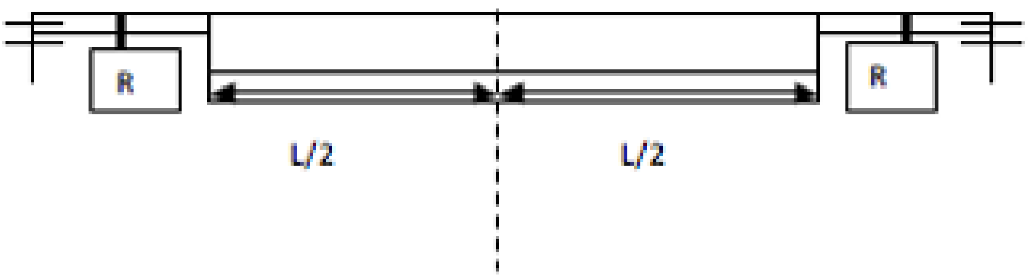

Figure 4.4: Resulting Pattern

4.1.1 METHODS TO ANALYSE A PATCH

As discussed earlier that a patch antenna can be treated as a two-dimensional planar structure and most of methods for analysis are based on this assumption.

Analytical Methods

1. Transmission Line Model

In the transmission line mode, the patch is viewed as a transmission line resonator with no transverse fields while the radiation mainly occurs due to the fringing fields at the open circuited ends.The effective dielectric constant is kept slightly less than the dielectric constant of the substrate so that the fields not entirely get confined to the substrate but also fringe and spread in the air. Here, two slots that are spaced by the length of the resonator represent the patch. Since the transmission line model does not take care of variation of field in the orthogonal direction to the direction of propagation, all types of configurations cannot be analyzed using this model. However, it is easy to use.

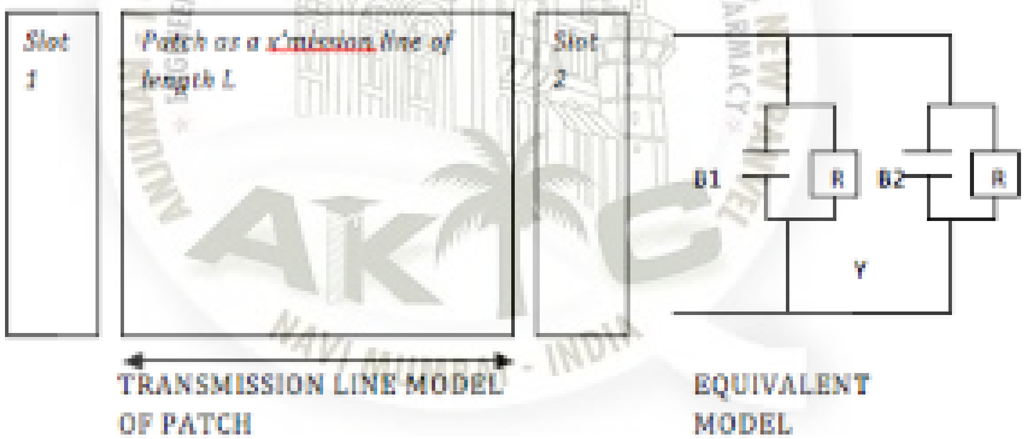

Figure 4.5: Transmission Line and its Equivalent Model

2.Cavity Model

Another method to analyse a patch is cavity model where the region between the patch and the ground plane is treated as a cavity, which is surrounded by the electric walls on the top and the bottom surface and the magnetic walls on the periphery. As the substrate used is thin as compared to wavelength inside the dielectric medium, the fields inside the cavity remain uniform along the thickness of the substrate while the equivalent magnetic current around the periphery computes the far fields and the radiation.

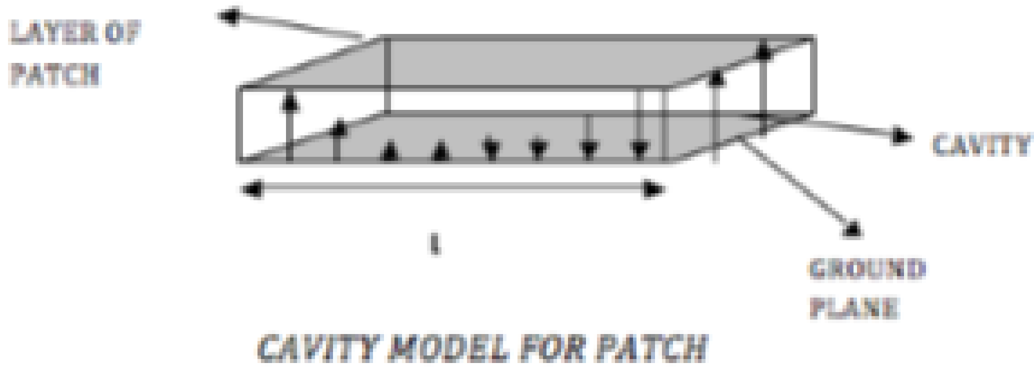

Figure 4.6: Cavity Model

3.MNM

This method models the electromagnetic fields underneath and outside the patch separately and the patch is analyzed as a two dimensional planar network with multiple ports around the periphery. By segmentation multiport impedance matrix is obtained and is solved. The far fields and radiation are obtained by the voltage distribution around the periphery. The above-mentioned analytical methods offer simplicity and physical insight however last two utilize corresponding voltage distribution obtain magnetic current distribution around the periphery. These methods work accurately for regular shaped patches but sometimes turn inaccurate when applied to a patch of arbitrarily shaped geometry.

Numerical Methods

1.Method of Moments

The method of moments is one of the most commonly used numerical techniques. Here the surface currents are used to model a patch. An integral equation is formed that is expanded in terms of some basis and testing functions and transformed into a matrix form that can be easily solved by a computer. This method takes into account effect of fringing fields so provides more exact solution.

2.Finite Element Method

Finite Element Method is suitable for volumetric configuration. In this method, the region of interest is divided into a number of finite surfaces. These discretized units may be any well-defined geometrical shapes such as triangles, tetrahedral etc. depending whether configuration is planar, 2D or 3D. This method is very handy when the shape of the patch is arbitrary. But its only limitation is that it needs a truncation boundary to be applied, which may result into some inaccuracies.

3.Finite Difference Time Domain

Finite Difference Time Domain method conveniently models patch antennas. It utilizes spatial as well as time grid for electric and magnetic fields over which solution is required. The entire domain is divided into small units called cells. The Maxwells equations in differential form are used in this method. The discrete time variations of the fields are determined at desired locations. Using line integral of electric fields the voltage across two locations can be obtained. The current is computed by using loop integral of magnetic field.

Chapter 5

Result

5.1 Microstrip Patch Antenna Model on HFSS

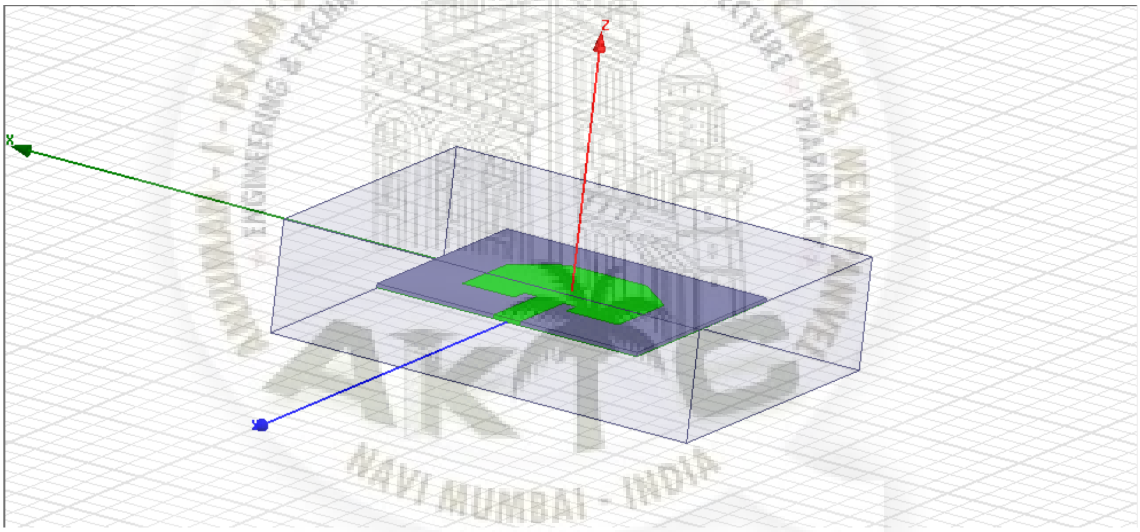

Figure 5.1: Side View

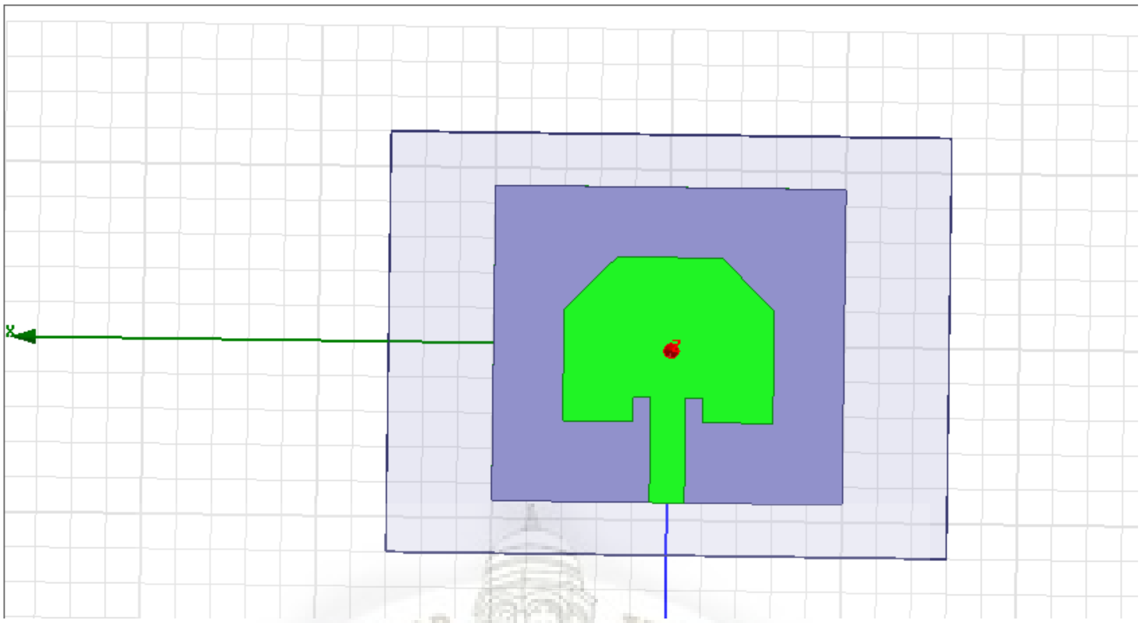

Figure 5.2: Top View

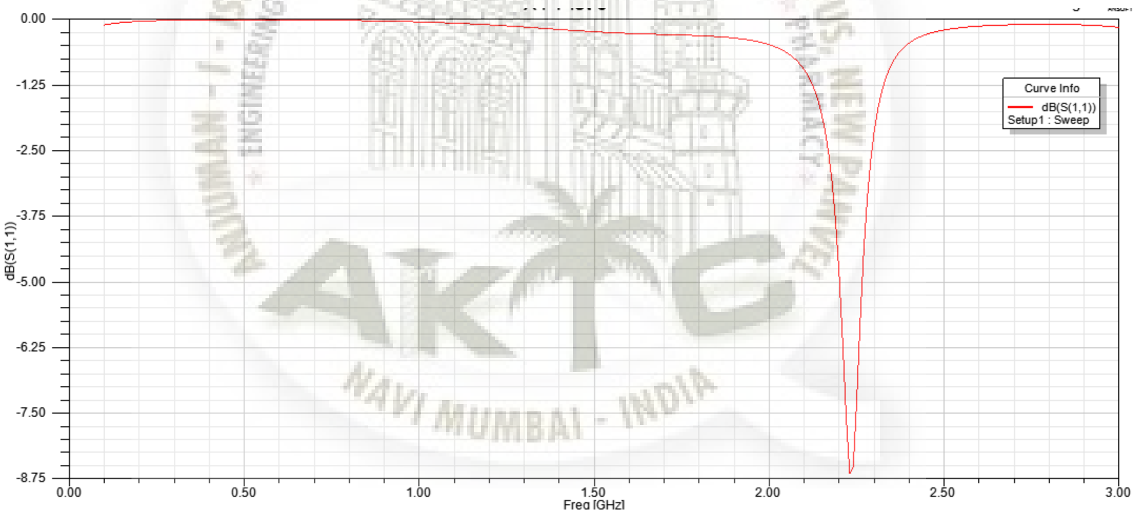

Figure 5.3: S11 Return Loss

5.2 Microstrip Patch Antenna Model on PCB

Figure 5.4: Microstrip Patch Antenna Testing using Vector Network Analyzers

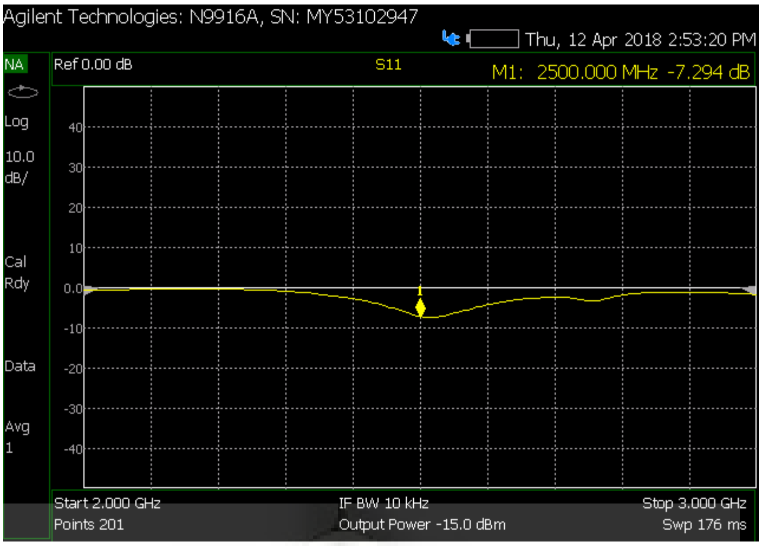

Figure 5.5: Return Loss

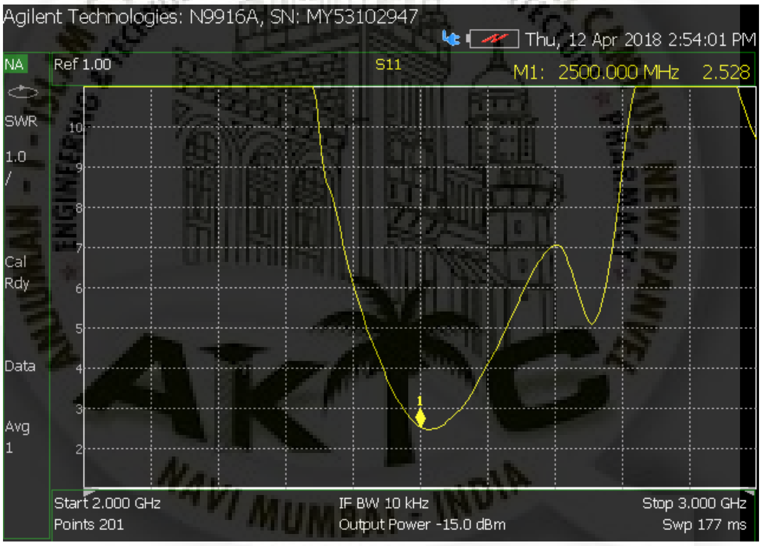

Figure 5.6: Return Loss

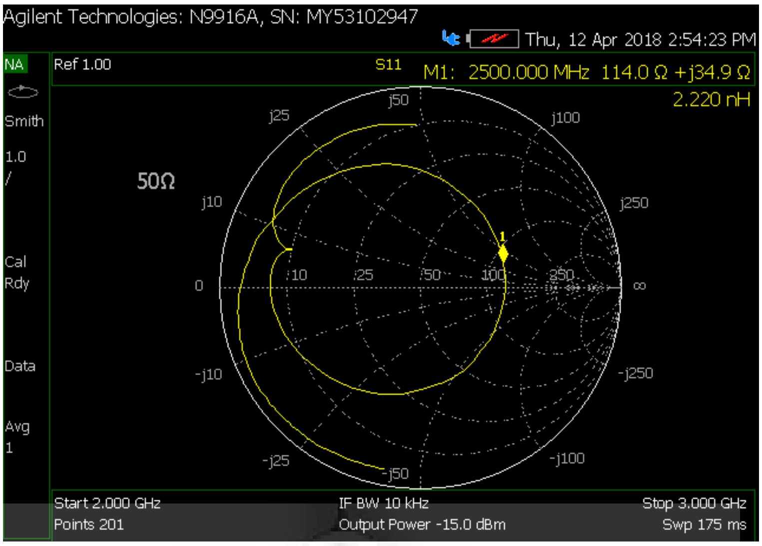

Figure 5.7: Radiation Pattern

5.3 Advantage

1.They operate at microwave frequencies where traditional antennas are not feasible to be designed.

2.This antenna type has smaller size and hence will provide small size end devices.

3.The microstrip based antennas are easily etched on any PCB and will also provide easy access for troubleshooting during design and development. This is due to the fact that microstrip pattern is visible and accessible from top. Hence they are easy to fabricate and comfortable on curved parts of the device. Hence it is easy to integrate them with MICs or MMICs.

4.As the patch antennas are fed along centerline to symmetry, it minimizes excitation of other undesired modes.

5.The microstrip patches of various shapes e.g. rectangular, square, triangular etc. are easily etched.

6.They have lower fabrication cost and hence they can be mass manufactured.

7.They are capable of supporting multiple frequency bands (dual, triple).

8.They support dual polarization types viz. linear and circular both.

9.They are light in weight.

10.They are robust when mounted on rigid surfaces of the devices.

5.4 Disadvantages

1.The spurious radiation exists in various microstrip based antennas such as microstrip patch antenna, microstrip slot antenna and printed dipole antenna.

2.It offers low efficiency due to dielectric losses and conductor losses.

3.It offers lower gain.

4.It has higher level of cross polarization radiation.

5.It has lower power handling capability.

6.It has inherently lower impedance bandwidth.

7.The microstrip antenna structure radiates from feeds and other junction points.

Chapter 6

Conclusion

6.1 Conclusion

The simulation of microstrip patch antenna is carried out. The various methodology necessary for simulation are done. The substrate and patches are created. The radiation pattern are observed. Rectangular patch antenna at 2.4 GHz is designed on ansoft HFSS. The designed antenna is suitable for Mobile Communication, Cell phone Antenna, etc. The HFSS can be used to small planar antenna topologies can be found in input impedance and radiation pattern. Few results are found about the efficiency. It is one of several commercial tool used for antenna design, and the design of complex RF electronic element including filters, transmission lines, and packaging.

6.2 Future scope

The explosive growth in the demand for wireless communication and information transfer using handsets and personal communications (PCS) devices has created the need for major advancements of antenna designs as a fundamental part of any wireless system. One type of antennas that fulfills most of the wireless systems requirements is the microstrip antennas. These antennas are widely used on base stations as well as handheld devices. Microstrip antennas have a variety of configurations and are currently the most active field in antenna research and development. The microstrip antennas, due to their great advantages, have increasingly wide range of applications in wireless communication systems as handheld mobile devices, satellite communication systems, and biomedical applications. In most PCS, the handheld antenna is placed on a small plastic/shielding box that is in close proximity to biological tissue of user body hence its radiation may cause health hazardous effects. Added to the operational requirements, the users and service providers usually demand wireless units with antennas that are small and compact, cost effective for manufacturability, low profile, and easy to integrate with other wireless communication system components. The antenna designer must consider all these issues besides the electrical characteristics of the antenna performance which include antenna tuning (operating frequency), VSWR and return loss (input impedance), bandwidth, gain and directivity, radiation pattern, diversity, and size of the chassis (expressed as a function of wavelengths) and specific absorption rate (SAR) of the antenna. These design considerations have led antenna designers to consider a wide variety of structures to meet the often conflicting needs for different applications.

This special issue contains different topics about microstrip antennas. New designs are investigated for several wireless communication applications. Papers are classified from survey about most literature publications in several topics as RF energy harvesting to new designs in UWB antennas, reconfigurable antennas, smart MIMO systems, and so forth.

Chapter 7

Reference

1. Shreema Manna,"Rectangular Microstrip Patch Antenna for Medical Applications"International Journal of Advanced Research in Electrical, Electronics and Instrumentation Engineering, Feb 2016.

2. Sonia C. Survase,Vidya V.Deshmukh,"Design of Wearable Antenna for Telemedicine Application " International Journal of Engineering Science and Innovative Technology (IJESIT),March 2013.

3. C.Vishnu Vardhana Reddy, Rahul Rana,"Design of Linearly Polarized Rectangular Micrstrip Patch Antenna Using IE3D/PSO ",IJAREEIE, May 2009.

4. James j., and P.S. Hall (Eds),"Handbook of microstrip antenna", Peter Peregrinus, London, UK,1989.

5. R.Garg, P.Bhartia, I.Bahl, A. Ittipiboon, "Micro strip Antenna Design Handbook",

ARTECH HOUSE, Boston2001.

6. David R. Jackson ,"Overview of Microstrip Antennas" Dept. of ECE University of Houston.

7. Alak Majumder,Design of an H-shaped Microstrip Patch Antenna for Bluetooth Applications, International Journal of Innovation and Applied Studies,Aug 2013.

8. Indrasen Singh, Dr. V.S. Tripathi,"Microstrip Patch Antenna and its Applications: a Survey" Motilal Nehru National Institute of Technology Allahabad.

7.1 Certificate

1.Participated in "International Level Paper Presentation" at Thakur College of Engineering and Technology.

Bibliography

[1] HM Srivastava, Shigeyoshi Owa, and Katsuyuki Nishimoto. Some fractional differintegral equations. *Journal of mathematical analysis and applications*, 106(2):360–366, 1985.