DOKUZ EYLÜL UNIVERSITY
GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

EMC DESIGN OF RF CIRCUITS

by
Gülşah GÖKER

November, 2012
İZMİR

EMC DESIGN OF RF CIRCUITS

A Thesis Submitted to the
Graduate School of Natural and Applied Sciences of DokuzEylülUniversity
In Partial Fulfillment of the Requirements for the Degree of Master of Science
in Electrical and Electronics Engineering Program

by
Gülşah GÖKER

November, 2012
İZMİR

M.Sc THESIS EXAMINATION RESULT FORM

We have read the thesis entitled **"EMC DESIGN OF RF CIRCUIT"** completed by **GÜLŞAH GÖKER** under supervision of **PROF. DR. HALDUN KARACA** and we certify that in our opinion it is fully adequate, in scope and in quality, as a thesis for the degree of Master of Science.

Prof. Dr. Haldun KARACA

Supervisor

Prof. Dr. Uğur Çam

(Jury Member)

Prof. Dr. Yalçın ÇEBİ

(Jury Member)

Prof. Dr. Mustafa SABUNCU

Director

Graduate School of Natural and Applied Sciences

ACKNOWLEDGEMENTS

First of all I would like to thank my advisor Prof.Dr. Haldun KARACA for his patience, kindness, guidance, support and separating his valuable time to support me.

I want to remark that my workplace Turkish Standards Institutionhave a great part to increase my experience especially on electromagnetic compatibility appliances and supply a good working place. I also want to thank my director of Electrotechnical laboratory A.Turan HOCAOĞLU and EMC department responsible Hilmi AKDOĞAN for encouraging me for achievement.

Finally I want to thank my mother Ayşe, father Osman, brotherGörkemOnur and sister Gizem who have always provided me tremendous help with support on my decisions and for keeping their infinite belief on me.

Gülşah GÖKER

EMC DESIGN OF RF CIRCUITS

ABSTRACT

In this thesis, the design criteria which shall be taken into consideration while designing radio frequency circuits are researched. So firstly the importance of the electromagnetic compatibility at high frequencies is emphasized. Precautions which can be taken against to the non ideal behaviors of circuit components at high frequencies are mentioned.

Designing and analyzing radio frequency circuit via computer simulation programs especially “Advanced Design System” software are represented with giving examples concerning the application.

Microstrip lines are considered on a preferential basis for observing and explaining the electromagnetic effects in “Advanced Design System” simulation program. Also the problems which may be encountered in practical appliances are brought out with realizing examples.

Keywords: Electromagnetic compatibility, high frequency effect, radio frequency circuits, Advanced Design System (ADS) software, microstrip line

RADYO FREKANS DEVRELERİNİN ELEKTROMANYETİK UYUMLULUK KONUSUNU DA DİKKATE ALARAK TASARIMI İÇİN ÖNEMLİ KRİTERLERİ İRDELEMEK

ÖZ

Bu tezde, radyo frekans devrelerini tasarlarken dikkat edilmesi gereken tasarım kriterleri incelenmektedir. Bunun için öncelikle yüksek frekans devrelerinde elektromanyetik uyumluluk konusunun önemine vurgu yapılmaktadır. Devre elemanlarının yüksek frekanslardaki ideal olmayan davranışlarına karşılık alınabilecek önlemlerden bahsedilmektedir.

Radyo frekans devresi tasarımı ve analizinin bilgisayar simülasyon programları aracılığıyla özellikle de "Advanced Design System" yazılımı ile nasıl gerçekleştirilebileceği uygulamalara yönelik örnekler verilerek sunulmuştur.

Elektromanyetik etkilerin "Advanced Design System"simülasyon programında gözlenebilmesi ve açıklanabilmesi için mikroşerit hatlar öncelikle incelenmiştir. Ayrıcamikroşerit hatlarla devre tasarımı örneklerle gerçekleştirilerek uygulamada karşılaşabilecek sorunlar göz önüne serilmektedir.

Anahtar sözcükler: Elektromanyetik uyumluluk, yüksek frekans etkisi, radyo frekans devleri, Advanced Design System (ADS) yazılımı, mikroşerit hat

CONTENTS

Page

CHAPTER ONE

INTRODUCTION

The subject of electromagnetic compatibility is rapidly increasing in importance along with the wide usage area of electromagnetic signals and digital electronics. As a result of transition from analog to digital, switching speeds got faster and electromagnetic emissions increased. However immunizing the electromagnetic susceptibility became more of an issue. Electromagnetic compatibility increasingly became a source of concern because of increasing interference with wire and radio communication.Reducing size of digital circuits has become obligatory with becoming more complicated of digital circuits. To keep electromagnetic interference in acceptable limits has become a serious effort with placing the circuit elements closer to each other. So it has been very important to design electronic systems for electromagnetic compatibility, especially for reducing the susceptibility. (Hubing, T., Orlandi, A., 2012)

However, governmental limits on the radiated and conducted interference of products play an essential role in terms of the importance of electromagnetic compatibility. These limits have been obligatory day by day due to restrict the electromagnetic noise emissions of digital electronic products. So that manufacturers and circuit designers have made a serious effort to comply the requirements of immunity and emission standards.

There are various kinds of computer simulation software that would help the design and compliance the requirements in respect of electromagnetic interference problem for RF circuit. Today computer simulation is an important part of designing circuits because of getting results of operating fast and correctly. Non ideal behavior of circuit components and connectors cause unintended consequences especially at high frequencies. It is possible to analysis the performances of circuit under ideal and non ideal operation conditions. Circuit simulators that take into account circuit layout, wiring techniques and physical structure of the circuit are imperative tools for circuit analysis and design.

Mostly used programs for RF circuit design are CST Microwave Studio Program and Agilent Advanced Design System program. CST Microwave Studio Program is a kind of electromagnetic field simulation software. In this thesis, Advanced Design System simulation software is studied on for designing and analyzing circuits in high frequency. There are mostly microwave and RF devices in library due to main scope of ADS is microwave and RF designs.

Design restrictions for digital circuits are taken into consideration to get accurate and reliable responses via software simulators. The facts of using mitigation methods for electromagnetic interference problems are notably effective. Therefore mitigation methods are detailed to understand the importance of all these methods. S-Parameter technique is used for characterizing radio frequency circuit and systems because of being simple, analytically convenient and easier to measure in high frequencies. Microstrip lines are used to simulate the electromagnetic effects that occurs due to transmission line, distances between the components or another effect as causing disturbances.

The aim of this study is to design and analysis the radio frequency circuit in Advanced Design System software with regarding the electromagnetic compatibility requirements. Hence the main issue of this research is to investigate the reasons of unintended electromagnetic effects and illustrate the importance of the effects of non ideal behavior of circuit elements, physical layout of circuit, operating frequency and parasitic of circuit components.

CHAPTER TWO

ELECTROMAGNETIC COMPATIBILITY (EMC)

The difference of national standards was in the forefront of problems in international commerce. Importers and exporters should have knowledge about different regulations and standards that change from one country to another. It was appreciated that this matter had caused many restraints in free circulation.Therefore, some renewals have been done in the scope of new approach directives to simplify international commerce.

In these days there are many technical requirements for free circulation of commodities in international commerce among European Community countries. These requirements are generally not for the quality of devices, they are mostly related to the subjects of safety and electromagnetic compatibility. Thus the quantity of standards has been reduced notably and that case has facilitated the international commerce of importers and exporters.

Electromagnetic Compatibility Directive (2004/108/EC) has been constituted with regulations concerned with electromagnetic requirements. This directive almost contains all electrical and battery operated devices. All of these devices that are in scope of this directive and supplied to the markets shall be marked with "CE" notation. In case of irregularity, there are many legal sanctions such as preventing commerce and recalling from market. Electrical device manufacturers may encounter commercial and legal issues at test process or later in case of not taking precautions for electromagnetic interference problems at design stage.

Electromagnetic compatibility is becoming more important as electromagnetic interference problems grow by the wide usage area of electromagnetic waves. In our daily life electromagnetic waves are used for almost everywhere such as communication systems, medical device technologies, defense systems and carrying voice and image signals. As the usage areas of these waves grow, it is ineluctable to confront with electromagnetic interference in systems which cause electromagnetic compatibility problems. Electromagnetic interference is undesirable energy that affects the performance of electrical/electronic equipment. The sources of electromagnetic interference are disturbance signals which are conducted and radiated.

In the view of this context electromagnetic compatibility should be defined formally from technical aspect. Electromagnetic compatibility is defined as"The ability of an equipment or system to function satisfactorily in its electromagnetic environment without introducing intolerable electromagnetic disturbance to anything in that environment." in the International Electrotechnical Vocabulary. Environment in this definition is "the totality of electromagnetic phenomena existing at a given location". To clarify the electromagnetic compatibility it is beneficial to mention about electromagnetic disturbance. Electromagnetic disturbance is defined in the International Electrotechnical Vocabulary as "Any electromagnetic phenomenon which might degrade the performance of a device, equipment or system, or adversely affect living or inert matter. An electromagnetic disturbance might be electromagnetic noise, an unwanted signal or a change in the propagation medium itself".

Electromagnetic compatibility is concerned with the interaction between equipments or circuits. It handles required arrangements for reducing the electromagnetic interference. The important parameters of the interference are source of interference (disturbance signal), device under the influence of interferenceand the transmission line by contacted or radiated. It should be acknowledged that undesired electromagnetic energy may be at anytime and anywhere in electromagnetic environment. So equipments should at least be affected by electromagnetic inherence. To provide this, international standard organizations prepare the required EMC standards and determine the limits of emission and immunity in these standards. To satisfy the specifications of EMC standards is very important for devices to operate properly without affecting each other. Maximum emission limits and minimum immunity limits should be regarded as designing circuits and systems.

It is advisable to taking care precautions of electromagnetic compatibility problems at the design phase for economic and temporal benefits.

There are some general techniques to reduce or eliminate the electromagnetic interference. As lined above, one of the matters that cause interference is the source of electromagnetic interference. It is one of the solutions to find the reason of interference's source and to eliminate it. Electromagnetic interference can be prevented with taking precautions after detecting the source of it. Another solution for eliminating the interference is to strengthen the device less susceptible to interference. In other words, electromagnetic immunity of devices should be increased to protect from undesired signals. So devices are reinforced against interference. Reinforced device has low probability to not operate properly. Eventual solution is about the transmission line by contacted or radiated. Coupling lines between devices should be optimized with regards to interference. There are many methods to prevent the transmission line from disruptive signals. Designing filters, shielding or grounding may be some of these methods.

Some terms which are concerned with the subject of electromagnetic compatibility are immunity, emission and magnetic susceptibility. Magnetic susceptibility and immunity are inversely proportional. Hence, low susceptibility means high immunity and vice versa. Magnetic susceptibility is a dimensionless proportionality constant that indicates the degree of magnetization of a material in response to an applied magnetic field. Electromagnetic immunity is the ability of a device to operate properly in the presence of electromagnetic environment. Devices should have high electromagnetic immunity in case not influence electromagnetic interference. Electromagnetic interference has an effect on devices by two ways. One of them is radiated the other is conducted. For instance, household and similar electrical appliance should have ability to function properly against radiated interference in the atmosphere especially in the frequency range of 30Mz to 1GHz. Radiated interference less than 30MHz frequency range is at the level that can be ignored. So radiated immunity of an appliance should be reinforced for the frequency range especially more than 30MHz. In the case of frequency is less than 30MHz, conducted immunity is at issue. Because at low frequency almost all the disruptive signals are radiated by conducted interference especially in the range of 150 kHz to 30MHz frequency range for household and similar electrical appliances. Mains-borne disturbance is the most common one among the conducted interferences. To achieve electromagnetic compatibility, devices should be able to be tested according to the requirements of immunity standards and meet pre-determined functional acceptance criteria. The last term which is concerned with the electromagnetic compatibility subject is electromagnetic emission. Electrical devices produce emissions as a result of their working. But emissions of a device should not affect another device's operation in the concept of electromagnetic compatibility. Hence, electromagnetic emission requirements must define the maximum allowable emission limits and testing procedures.

If devices are suitable to operate in the same frequency range, they should be kept away from each other for not causing electromagnetic interference. In the case of having to operate closely, then there is one solution to not influence each other. It is only possible with operating in different frequency range. This matter is also valid for the components which have to operate in a circuit in the same frequency range. Some precautions shall be taken to prevent the electromagnetic interference in the circuit, so it is provided that the components fulfill the requirements of electromagnetic compatibility. There are many ways to satisfy the conditions that the components not affect each other's operation.

As mentioned before, the EMC standards determine the limits of emission and immunity. The requirements of the standards are for controlling the amount of electromagnetic interference produced by products. By this way the electromagnetic interference is minimized. If there are rules like the requirements of the standards which are mandated by governmental agencies, manufacturer must obey these rules in order for the product to be sold in a country. Besides obeying the rules of governmental agencies, the requirements of manufacturer's self shall be complied. It is possible to make product to comply with the susceptibility requirements. Most of disturbances of electrical devices originate from conducted emissions. Because of that replacing the ac power cord with internal batteries can be a solution for conducted interference. As another example, if the product is affected by another product's interference on radiated emissions at a particular frequency, it should be made to comply with the limits. The first step to better the situation is determining the source of the emission. It is important so that the most suitable solution can be generated before arising bigger problems. For complying with the electromagnetic compatibility the product should be isolated from surrounding area in respect to destructive interference. The strongest effect is done by the cables which are stated next to product, because cables perform a duty of efficient antennas for radiation of high frequency signals. The most reliable way to reduce the radiated and conducted interference of a product is to add suppression components that reduce the levels of emissions. However, these components add extra cost to the product over that required for its functional performance. Increase of price is an un-solicited status because the cost is the primary consideration in respect to manufacturer. In past years the most important consideration was the quality of product. But nowadays this mentality is not valid. Today's primary consideration is to create optimum product in the view of the fact that lower cost for mass production.

2.1Aspects of Electromagnetic Interference

Solutions generated for electromagnetic problems varied by increasing of electromagnetic interference problems. Before mentioning the solutions, all the problems about the subject of electromagnetic compatibility shall be intellectualized and all the limits determined for not causing the electromagnetic pollution shall be explicated.

It is necessary to design electronic circuits and systems with taking into consideration the requirements of electromagnetic compatibility. Devices operate properly in the presence of electromagnetic signals which are under the determined limits. However, on the contrary namely, if the electromagnetic signals are out of the determined limits, devices cause interference with another device or are influenced from other devices because of having low immunity. It is possible to draw a conclusion from these cases that not only EMC design techniques are important but also meeting the requirements is so important.

System or device shall not cause interference with other systems or devices. They shall also have low susceptibility and high immunity to emissions from other systems. In the case of meeting these requirements, it is possible to say system or devices meet the requirements of electromagnetic compatibility.

As it is understood from the mentioned requirements, electromagnetic compatibility internal to a system is a different subject from electromagnetic compatibility interactions between the systems. A general aspect about the interaction of systems is shown in the below Figure 2.1. This figure illustrates a basic view of electromagnetic design.

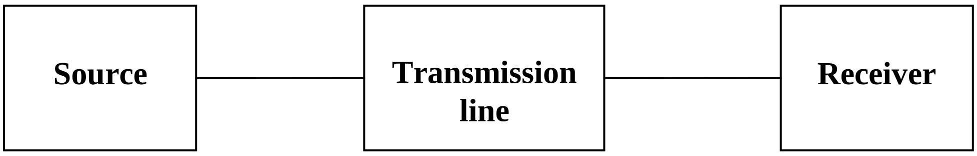


Figure 2.1 A general aspect of electromagnetic design

Electromagnetic signals emits from source to receiver. These signals follow a path during emitting. Path can be described as transmission line in this interaction. Source produces emissions. Transmission line transfers these emission signals to receiver by conducted or radiated path. Emission signals are not always the disruptive signals; they can also be desired signals. If the signals are disruptive, causing interference on receiver is one of the possibilities. Therefore, the receiver could behave in an undesired manner. From this process, it is realized that the emitted signal determines the importance of interference. Transmission path is also matter as well as source. Electromagnetic compatibility problems originate from four main items; radiated emissions, radiated susceptibility, conducted emissions and conducted susceptibility. For reducing the conducted interference, length of cable shall be shortened. Long cables may behave as an antenna and collects all radiated emissions from environment. Then, these signals may be transferred to inside the systems and cause the electromagnetic interference. Usually conducted line is more efficient than radiated line.

The importance of transmission path shall be reduced to prevent the electromagnetic interference. Placing the barriers, filters in the circuit prevent the undesired signals. Reducing the frequency of emitted signals makes less efficient the transmission path. Another way to prevent interference is using shielded enclosures for receivers or using internal batteries instead of mains supply. But these ways will increase the cost. While designing the system in the sense of electromagnetic compatibility, minimizing the cost shall not be ruled out. There are also some ways about source and receiver to prevent interference. Undesired signals emitted from the source shall be suppressed as possible. Receiver shall be made less susceptible to emissions transmitting from source. Electromagnetic compatibility subject is concerned with the ability of electromagnetic emissions to cause interference in electrical and electronic devices.

The spectral content of digital devices generally occupies a wide range of frequencies and can also cause interference in electrical and electronic devices. For example, a strong transmission from an FM radio station or TV station may be picked up by a digital computer, causing the computer to interpret it as data or a control signal resulting in incorrect function of the computer. Conversely, a digital computer may create emissions that couple into a TV, causing interference. (Paul, C.R. ,2006)

To summarized, electromagnetic interference problems may be emerged as designing the circuits or systems. However these interferences shall be prevented by efficient precautions in the process of design.

2.2Design Restrictions for Digital Circuits

The electromagnetic compatibility requirements imposed by governmental agencies shall be considered for controlling the electromagnetic noise generated by products. It is so important to obey the requirements for devices function properly. It has also importance that determining the source of emissions in design stage for avoiding extra costs that may occur in next stages.

Fixing the problems in time obstruct occurring new problems. Thereby determining the interference problems and emission source minimize the cost of manufacturer. After the product produced, solving the electromagnetic problems with protective methods is becoming more difficult and expensive. Main methods used in protecting the circuit from interference are grounding, shielding, filtering, cables and connectors.

2.3 Mitigation Methods for Electromagnetic Compatibility Problems

2.3.1 Grounding

Ground is defined as a direct electrical connection to the earth or a connection to a particular point in an electrical or electronic circuit. But connection can also be by an indirect way which operates as the response of capacitance between wireless equipment. Grounding is defined as “to connect the electrical circuit or equipment to the earth potential or a reference surface with a size that stands for earth potential via interconnections” in IEEE (Institute of Electrical and Electronic Engineers).

Grounding methods which are used in general electrical systems are also valid for protection against electromagnetic interference. Grounding that is done by suitable methods, has importance for reducing the electromagnetic interference problems. Some basic problems of electromagnetic interference can be solved by grounding without using filter systems which have high cost.

The most important problem about grounding is that groundings done in different points have different potentials. The reason of this is non-zero impedance of earth. It is zero in ideal situations but it is not possible in practical situations. Non-zero impedance between different points causes difference in voltage.

There are many ways for grounding. Grounding methods change according to connection types of different components or modules. Which type of grounding used is related to the operating frequency range. Analog circuits which have low operating frequency are suitable for single point grounding.

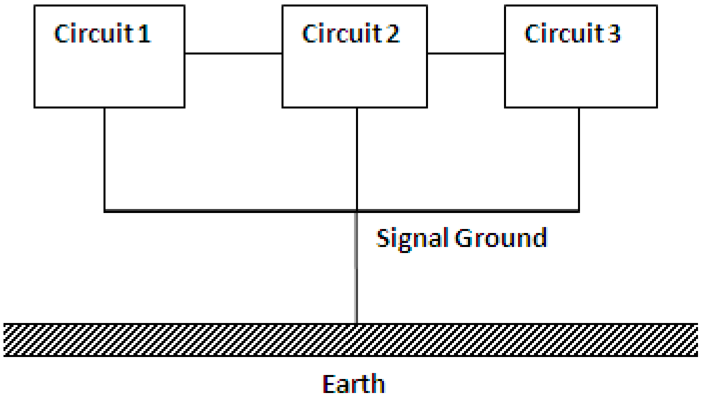


Figure 2.2 Single point grounding configuration

Conversely, multi point grounding method is used in circuits which have high operating frequency. It is very important to do grounding well in printed circuits. There are many points to consider doing grounding well. Pins of components should be short and twisted pair lines should be used. Main signal lines should be next to reference point. Location of circuit components is very important to reduce the inductance effects and potential difference in circuit. (Sevgi, L., 2006, Joffe, E.B., Lock, K. ,2010)

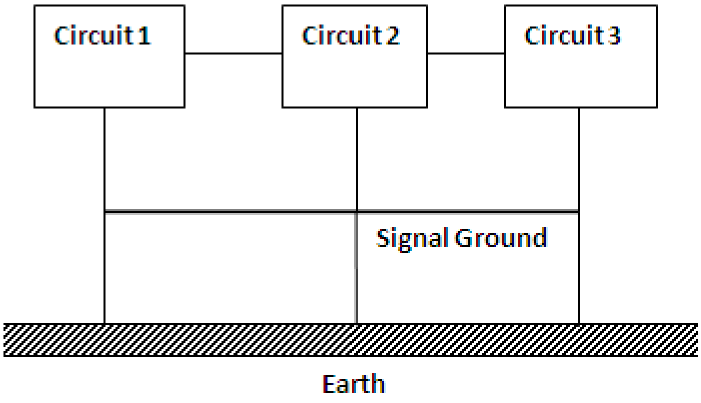


Figure 2.3 Multi point grounding configuration

2.3.2 Shielding

Electromagnetic shielding is defined as a housing, screen or other object, usually conducting, that substantially reduces the effect of electric or magnetic fields on one side thereof, upon devices or circuits on the other side. (Celozzi, S., Araneo, R., Lovat, G. , 2008)

Shielding is done for preventing the entrance of electromagnetic signals to the defined region. So, it is one of the main methods to isolate regions from each other with regards to electromagnetic fields. There are two kinds of shielding; electrical shielding and magnetic shielding. If there is a strong electric field around the source of interference, the method that is a solution for interference should be electrical shielding. Magnetic shielding should be used in the case of presence of strong magnetic field around the interference of source. Electrical shielding is provided with material which has a good conductance. For example stainless steel, copper and conductive metal are among the materials that have high performance for electrical shielding. Otherwise, magnetic shielding is provided with filter which is comprised of ferromagnetic material. (Gonschorek, K., Vick, R., 2009)

Shielding provides electric field to be reflected in low frequencies and to be absorbed in high frequencies. Shielding provides magnetic field to be absorbed in low frequencies. High conductivity effects reflection and absorption positively. High magnetic conductivity causes high absorption and low reflection. In case of shielding supplies with very low frequency, materials with high magnetic conductivity are used. For magnetic fields thick shield are used, for electric field thin materials are used. The distance between the source and shield changes reflection characteristics. Electric sources should be located next to the shield, magnetic sources should be located far from the shield. (Paul, C.R. ,2006)

In practical applications magnetic field shielding is important in low frequencies beside electrical field shielding is important in high frequencies. There are many factors that affect the quality of shielding. Some of them are internal resistance of shielding, holes, openings and specifications of shielding materials. Connections and solder joints are among the main factors that affect the quality of shielding negatively. If the cable connections used in circuits or systems are inherently shielded such as coaxial cable, efficiency of shielding is mostly good. If cable connections are open connections and not inherently shielded such as microstrip lines, efficiency of shielding should be considered more because performance may be not sufficient. (Sevgi, L., 2006, Paul, C.R., 2006)

2.3.3 Filtering

Filtering is one the solutions of electromagnetic interference problems. A simple filter consists of capacitive and inductance components. Noise could be reduced or reflected by filtering. Filters bloke undesired signals at determined frequencies. They are also used for allowing signals in intended frequency range.

Filters are classified in four groups. They are low pass filter, high pass filter, band pass filter and band elimination filter. High frequency filters block or reduce the signals under 50Hz. They allows to the high frequencies. Band pass filters allow the signals in a determined frequency range, but prevent the signals expect this range. Band elimination filters are designed for blocking noise signals in specified frequency range. They are especially used in the circuits used for receiver and transmitter. (Arı, N., Özen, Ş., 2008, Gonschorek, K.,Vick, R., 2009)

The interference frequency is generally above from the wanted signal. So low pass filter is the most common filter among them due to reducing the noise in high frequency and radio frequency. It consists of two components; inductance or resistance in serial and capacitor in parallel.

The choice of inductance or resistance for the series element is usually determined by the intended signal current that has to be passed: power filters will typically be unable to stand more than a few ohms resistance, but signal filters might easily be able to cope with kilo ohms. Resistance has the advantage that it absorbs interference energy and does not contribute to resonances, and is of course cheaper and smaller than inductance. Inductance on the other hand can provide high RF impedance with little DC or low frequency loss.(Williams, T., Armstrong, K., 2000)

Noise occurred from the mains is one of the most common source of electromagnetic interference. Mains filters are used to solve this problem. They are placed into the entrance of mains power supply or on shielding. There are some points to take care when choosing the filters. Function specifications such as voltage, current, temperature effect, operating frequency, electrical specifications such as high voltage characteristics, isolation resistance, mechanical specifications such as dimensions and placement in circuit are very important for designing circuit. Placement on the shielding has to be considered for the performance of filtering. Filter shall be placed on shielded mounting surface in the entrance point of power supply cable for good performance.

Cables behave as antennas in high frequencies. Filters made by ferrite are used in a circle form to reduce the electromagnetic interference arise from wiring cables and signal paths. Ferrite filters are also easy to use because of its practical use. They also have a wide usage area for telecommunication devices. They filter the unwanted signals in different frequency ranges. They have no effect on wanted signals and low frequency signals. Construction and shape of ferrite filters are selected for better performance. (Sevgi, L., 2006, Morrison, R., 2002)

When selecting a ferrite for highest impedance, longer filter is better than fatter in a viewpoint of shape and construction. The impedance for a given core material is proportional to the log of the ratio of outside to inside diameter but directly proportional to length. This means that for a certain volume (and weight) of ferrite, best performance will be obtained if the inside diameter fits the cable sheath snugly, and if the sleeve is made as long as possible. A string of sleeves is perfectly acceptable and will increase the impedance pro rata, though the law of diminishing returns sets in with respect to the attenuation.(Williams, T., Armstrong, K., 2000)

2.3.4 Cables and Connectors

Cables are one of the main factors that affect the performance of electromagnetic compatibility. It is so important to design circuits with regarding the requirements of electromagnetic compatibility in the view of cable and connectors. They have the largest area in a circuit. Interference signals are transmitted from one point to the other in a circuit via the cables. Cables also behave as an antenna in high frequencies as mentioned in previous part. So they can be called as the source of electrical and magnetic interference. Beside the quality of cable and material, coupling type, cable shielding and method of grounding are the fundamental criterions that have importance on electromagnetic compatibility performance.

Inherently shielded cables such as coaxial cables shall be used for better performance. If it is not possible to use cables which are inherently shielded, cables shall be shielded against the electrical and magnetic interferences. In practical use while copper provides electrical protection, ferrite material provides magnetic protection. So they shall be used together for cable shielding.

Shielding has importance on cables because of preventing the noise in the circuits especially in high frequencies. Unshielded cables show inadequate performance with regard to occur noise voltages as frequency increases. If shielded cables are used, it is not possible to increase of noise voltage as frequency increases. Shielding is effective upper the cut off frequency not in lower frequencies.

There is also one more point to consider in the usage of cables in the view of electromagnetic compatibility performance. It is the method of grounding. In high frequencies it is not enough to make connection to ground only at the side of load. Electrical protection is provided by one point grounding. Grounding in two points which are at the sides of source and load helps to providing magnetic protection. Grounding shall be made by wide tapes. Cable connections shall be contacted firmly to the surface. Connections shall also be protective against to the electrolytic effects. (Sevgi, L., 2006, Gonschorek, K., Vick, R., 2009)

2.4Behaviors of Circuit Components at High Frequency

Basis components such as resistor, inductance, capacitors have different behavioral in circuit design concerned with the gain of usage. It is a good case to study on especially in radio frequency, microwave and all other types of high frequency digital circuits. Their physical characteristics lead them to behave different in different frequencies especially in high frequency applications. For example a capacitor element that functions as a capacitor at 100 MHz may behave as an inductor above 1 GHz frequency. Analyzing the frequency reactions of these components help to solve problems in electromagnetic compatibility. But their ideal behaviors are not the same with in application. Not only the non ideal behaviors of components are important, but also the non ideal behavior of system should be considered.

2.4.1 Wires and Component Leads

Some components such as resistor and inductance contain wire with determined shape and length. Behavior of conductors changes according to the diameter and length of wires. Materials, conductivity, permeability and shape of wires are parameters affect the occurred interference in the circuit.

The resistance of the conductors is generally more important in the functional design as in determining the required land size and/or wire gauge to ensure minimum voltage drop along them in a power distribution circuit. However, at the frequencies of the regulatory limits and particularly in the radiated emission range the inductance of the conductors is considerably more important than the resistance. (Paul, C.R. ,2006)

Lead connections are as important as wire connections. They also have a wide space in a circuit. A component is connected to the circuit via lead connections. There are many points to consider as attaching the components to circuit lands for example printed circuit boards.

One of the most important factors that affect the high-frequency behavior of components is the length of the component attachment leads. Unnecessarily long attachment leads cause the component behavior to deviate from the ideal at high frequencies, which often fall in the frequency range of the regulatory limits where we want the component to behave as expected. The length and separation of the component leads cause the component to have, in addition to the ideal behavior, an inductive element and a capacitive element. These elements in combination with the component can give an overall behavior that is far from the desired ideal behavior. (Paul, C.R. ,2006)

2.4.2 Resistors

Resistors are the most common components in circuits. Ideally a resistor has a magnitude equal to the value of resistor.

At low frequencies such as other components resistors behave nearly identical to the ideal behavior. However at high frequencies their behaviors change according to frequency response. Components exhibit unwanted characteristics that are called parasitics.

There are three types of resistor which are called metal film, carbon composition and wire wound. Carbon composition has good transient immunity, medium precision and easy fabrication. Besides having these good features they are not durable to high currents. On the other hand wire wound resistors have high power and inductance. But they are more expensive than carbon resistors.

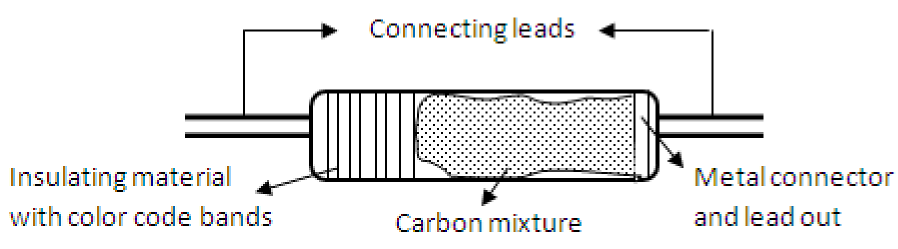


Figure 2.4 Internal structure of a carbon composition resistor.

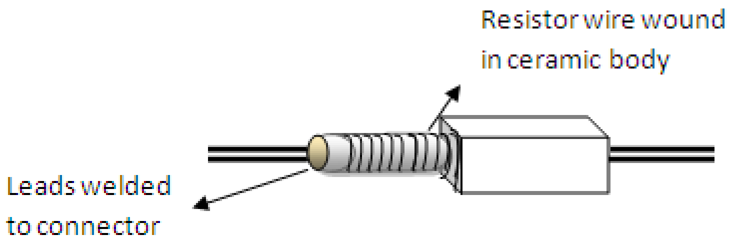


Figure 2.5 Internal structure of a wire wound resistor.

Metal film resistor is constructed of a thin metal film on an insulating layer. They have high precision and medium inductance with a comparison to the other types of resistors.

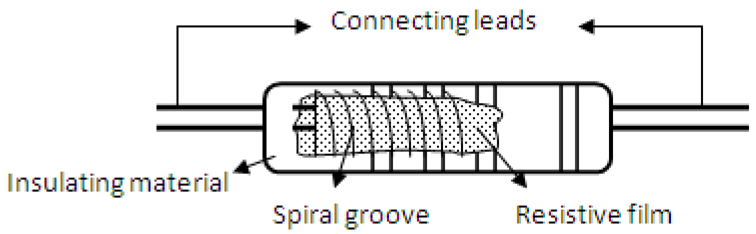


Figure 2.6 Internal structure of a metal film resistor.

It does not matter which type the resistor is. All of the types have different frequency responses because they are all frequency dependent. An equivalent circuit schematic of non ideal resistor is represented in below figure.

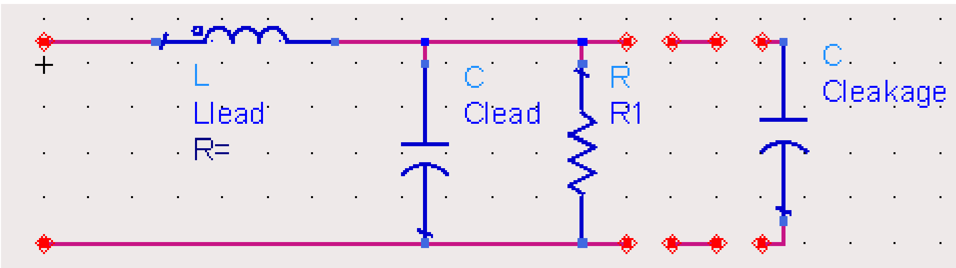


Figure 2.7 Equivalent circuit of resistor in high frequency range. (Paul, C.R. ,2006)

Capacitor called as Clead is in a parallel combination with resistance. Reason to occur Clead and Llead components are the leads of resistor. Llead is attached to resistor in a serial form. These are all parasitic effects of high frequency response. Clead and Cleakage constitute capacitor of parasitic effect. So it is possible to represent them in one component called Cparasitic.

Parasitic effects limit the frequency range of the resistor. As frequency increases, the impedance of the equivalent circuit decreases until to resonate. The equivalent circuit impedance has minimum value at the self resonant frequency of the resistor. While the impedance of lead inductor gets bigger, on the contrary the impedance of the parasitic capacitor gets smaller. Even it approaches zero. Hence resistor takes the form of open circuit schematic. To achieve a better performance and reduce the parasitic effect, the leads of resistor shall be as short as possible. (Paul, C.R. ,2006)

2.4.3 Capacitors

Capacitors are the components which have different frequency responses in high frequencies such as resistors mentioned before. Mica, ceramic, tantalum and other electrolytic are the types of capacitor. Mica type is used for high voltage applications. Tantalum and ceramic are the most used types in electromagnetic compatibility. Tantalum types are polarized and have good energy density. Ceramic ones are more stable and have better precision. They also have smaller capacitance than tantalum types. It is important to choose the right type of capacitor while designing circuit. Main factor is the operating frequency of capacitor. (Hubing, T.H., n.d.)

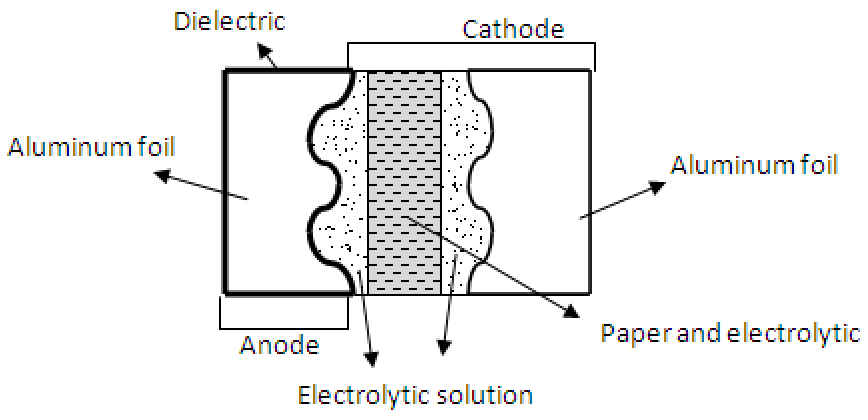


Figure 2.8 Construction of an aluminum electrolytic capacitor.

Ideally capacitors have a reactance impedance of Zc=-j/2πfC. There is a dielectric material between the plates of capacitors. Capacitors have inductance and resistance. In high frequency applications parasitic effects occur. An equivalent circuit schematic of physical capacitor is represented in below figure.

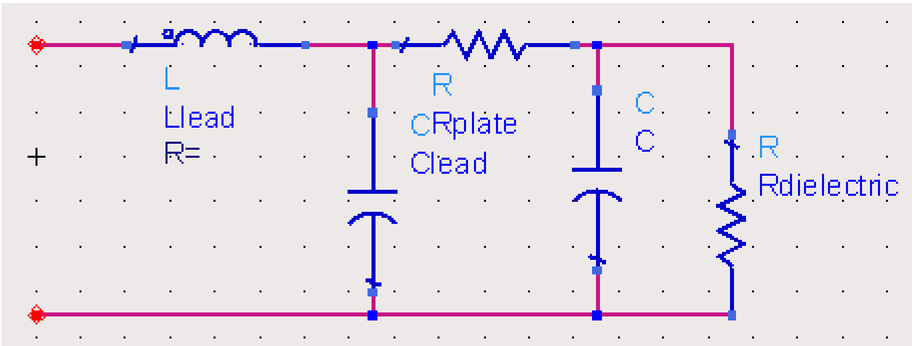


Figure 2.9 Equivalent circuit of capacitor in high frequency range. (Paul, C.R. ,2006)

Llead and Clead are the response of capacitor's parasitic effects in high frequency. Clead may be ignored because of being much less than C. Llead represents the lumped inductance of the leads. Rdielectric is the parallel resistance in dielectric with a big value. Rplate is smaller than Rdielectric.

Lead lengths are also important factors to constitute parasitic effects. As frequency increases above the resonant frequency, the impedance of capacitor increases too. Behavior of capacitor changes above the frequency of resonant frequency. Capacitors behave as inductors and lead inductance behave as open circuit. The operating frequency is determined by the leads of capacitors and the inductance of capacitor components. (Paul, C.R. ,2006)

2.4.4 Inductors

Ideally inductor has the impedance Z_L=j2πfL, which is purely reactive and has a phase angle of +90°for all frequencies. The construction of inductors is explained as wires roll on the coil. The material which the wires rolled on may be non magnetic or magnetic material. Magnetic materials are usually made of ferrite. Air core, ferrite core, iron core and laminated core inductors are the types of inductors. They are all classified according to the material of core. Air core inductors act linear behavior under high current conditions. But these types of inductors are the most susceptible to noise signals and have more effect on causing unwanted signals. Ferrite core inductors have high inductance. (Paul, C.R. ,2006)

In high frequency applications, inductors have different frequency responses. An equivalent circuit representation for physical inductor is shown below.

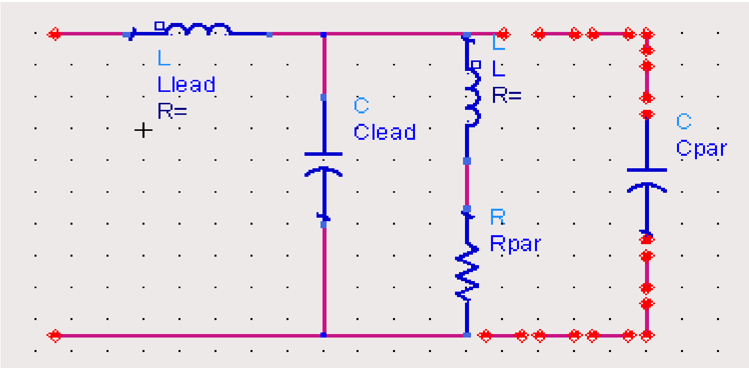


Figure 2.10 Equivalent circuit of inductor in high frequency range.
(Paul, C.R. ,2006)

Resonance of inductor and magnetic saturation are the two factors which occur as a response of non ideal inductor. Llead is the inductance in serial which is emerged by the reason of leads, Clead is the capacitance in parallel of leads as if Llead. Rpar is in serial with the ideal inductance and it is the self resistance of the windings. Lastly, Cpar is the parasitic capacitance which is in parallel form. Llead is much smaller than the values of L and Rpar so it may be ignored. In the same view, Cpar is much bigger than Clead.

The impedance of the equivalent circuit takes the maximum value at self resonant frequency. If the frequency goes on to increase above this frequency, the impedance of inductor decreases thus inductor behaves as if capacitor. This is not a wanted result. So some improvements shall be done to delay this case. Leads of inductor shall be shortened and Cpar shall be reduced. Using small value of inductance is advisable for high frequency applications.

As a conclusion, basic circuit elements are mentioned that they have parasitic behavior at high frequencies. But it is not limited with only these components;

resistors, leads, conductors and inductors. It can be said that all components in a circuit have parasitic behavior such as diodes, transistors and etc. As it is said many times before selecting the components with small values reduce the parasitic effects at high frequencies. Thus the performance of circuit increases via these improvements. Finally, it is also important to select the right type of component. Operating frequency of component shall be taken into consideration as designing the circuit.

CHAPTER THREE

RF CIRCUIT DESIGN AND ANALYSIS TECHNIQUES USING COMPUTER SIMULATION

There are various kinds of computer simulation software for RF circuit simulations available. Today computer simulation is an important part of designing circuits because of getting results of operating fast and correctly. It is possible to see the function performances of circuit under ideal and non ideal operation conditions. It can be checked how the circuit performs for all the component tolerances if possible. Spice models are usually used in computer simulation software for obtaining accurate and reliable simulation consequences. For RF circuit simulation it is so important computer simulation programs to have all relevant components function as linear or nonlinear models. RF circuit simulation programs are especially design to perform the effects of high frequency and used with the form of transmission lines. Today there are many computer software commercial of free that would help the design in respect of electromagnetic compatibility problem.

CST Microwave Studio Program is a kind of electromagnetic field simulation software. It is possible to design and analysis circuits in high frequency. There are many choices for simulation techniques such as transient, frequency domain integral equation, multilayer, asymptotic and eigenmode solvers. In the case of confronting different simulation problems, there are many methods to handle that are named solvers. The program especially suited to the fast, efficient analysis and design of components like antennas (including arrays), filters, transmission lines, couplers, connectors (single and multiple pin), printed circuit boards, resonators and any more. (Computer Simulation Technology AG, 2012)

Besides CST Microwave Studio Program, there is another program to mentioned about and study on it. Advanced Design System is used in electronic design automation software for RF, microwave, and high speed digital designs. It is possible to develop all types of RF designs, from simple to the most complex, from RF/microwave modules to integrated MMICs for communications and aerospace/defense applications. With a complete set of simulation technologies ranging from frequency and time-domain circuit simulation to electromagnetic field simulation, ADS lets designers characterize and optimize designs. (Agilent Technologies, 2012)

At first sight it is a bit complex circuit simulator but simply ADS software has a similar usefulness as other SPICE programs like PSPICE. While studying on it step by step it will be clear that one can build a design, set up the simulation, plot and display the results of the simulation. Circuit schematics can be drawn by means of graphical user interface with choosing components from predefined libraries. There are mostly microwave and RF devices in library due to main scope of ADS is microwave and RF designs. Various projects can be created, simulated, analyzed and optimized on ADS.

3.1 Simulation

There are many different simulation types (DC analysis, AC analysis, Harmonic Balance, S-Parameter analysis, Transient analysis, etc.). Among these simulation types S-Parameter Analysis is the most used one in ADS. Because the focus of ADS is RF and microwave design. S-Parameter Analysis will be especially emphasized in next sections. The analyses used for simulations in ADS software are as follows:

3.1.1 DC Analysis

The DC Analysis uses a system of nonlinear ordinary differential equations to solve for an equilibrium point in the linear/nonlinear algebraic equations. Instead of open circuits capacitors and similar items are replaced and instead of short circuits inductors and similar items are replaced. Besides independent sources are constant valued, time-derivatives are also constant (zero). In this analysis linear elements are replaced by their conductance at zero frequency.

The DC controller is used when applying DC analysis. After DC simulation DC controller displays voltages, currents and determines power consumption of the circuit.

3.1.2 Transient Analysis

Transient Analysis is used to solve nonlinear circuit in the time domain. It is a numerical algorithm that uses integration method, is used to replace the time derivative with a discrete-time approximation. The transient controller solves a set of integro-differential equations into algebraic equations that indicates the time dependence of circuit currents and voltages. By means of convolution or a simplified equivalent-circuit model linear components can be simulated in this analysis.

3.1.3 Harmonic Balance

Harmonic Balance is applied in frequency domain so that it is more accurate at solving high frequencies. By means of Harmonic Balance system of nonlinear ordinary differential equations become a system of nonlinear algebraic equations. Harmonic balance is faster than transient analysis at solving typical high-frequency problems because of cannot be accurately solved or can be solved at high prices with transient analysis. Because of being frequency-domain analysis technique, Harmonic Balance can simulate distortion in nonlinear circuits and systems. Harmonic Balance controller is used for performing nonlinear noise analysis and simulating analog RF and microwave circuits. It determines the spectral content of voltages or currents.

3.1.4 AC Analysis

For AC Analysis after the DC operating point is found, the nonlinear devices are linearized around that point. Small-signal AC simulation is also performed before a harmonic-balance (spectral) simulation to generate an initial guess at the final solution. AC Controller is used to obtain small-signal transfer parameters like voltage gain, current gain, transimpedance, transadmittance and linear noise. (Agilent Technologies, 2012).

3.1.5 S-Parameter Analysis

Linear networks and components may be described in the frequency domain using Scattering parameters. Using S-Parameters, common electrical characteristics including gain, return loss and insertion loss can be explicated in S-Parameter model. Due to simplicity this model is very practical in use. S-Parameter Analysis is the most used simulation type and besides being a type of small-signal AC simulation, it is the microwave equivalent of AC analysis. By means of this simulation it is possible to calculate the S-Parameters of multiple N-port networks in a single simulation run. S-Parameter Controller is used for simulating group delay, linear noise and the effects of frequency conversion on small-signal. It is easy to get S-Parameters of a circuit and convert them to Y- or Z-parameters via this controller. S-Parameters use normalized incident and reflected traveling waves in each network port. The signal-wave response of an n-port electrical element can be defined at given frequency. It is also possible to characterize RF component and arrange the specifications of a circuit element like temperature, bias, etc.

If the circuit contains any nonlinear devices, a DC simulation is performed first. Following the DC bias simulation, the simulator linearizes all nonlinear devices about their bias points. A linearized model captures the small incremental changes of current due to small incremental changes of voltage. These are derivatives of the transistor model equations, which are evaluated at the DC bias point. Nonlinear resistors and current sources are replaced by linear resistors whose values are set by the small signal conductance dI/dV. Current sources that depend on voltages other than the voltage across the source are replaced by linear dependent current sources dI1/dV2. Nonlinear capacitors are replaced by linear capacitors of value dQ/dV. The linear circuit that results is analyzed as a multiport device. Each port is excited in sequence, a linear small-signal simulation is performed, and the response is measured at all ports in the circuit. That response is then converted into S-Parameter data, which are in turn sent to the dataset. S-Parameter simulation normally considers only the source frequency in a noise analysis. (Agilent Technologies, 2012)

Showing examples in Advanced Design System program will help increasing the understanding of how it is used. In the Chapter Five, circuit simulation using ADS software will be clarified. Before explaining how simulation is done in ADS software,in next chapter S-Parameters will be explained.

CHAPTER FOUR
CHARACTERISING RF CIRCUIT AND SYSTEMS USING S-PARAMETER TECHNIQUE

S-Parameters are the most preferred method in microwave design because besides being simple, analytically convenient they are easier to measure and to work with at high frequencies than other kinds of two-port parameters. It is possible to understand S-Parameters better by firstly having more knowledge about transmission line theory.

At very low frequencies, with wavelengths much larger than the line, the transmission line can be thought of a simple wire. This is adequate for conducting DC or very low frequency power. The resistance of the wire is relatively low and has little effect on low-frequency signals. The voltage and current are the same no matter where a measurement is made on the wire. At higher frequencies, wavelengths are comparable to or smaller than the length of the transmission line (or a conductor in a high-frequency circuit), and power transmission can be thought of in terms of traveling waves. (Sischka, F. 2002)

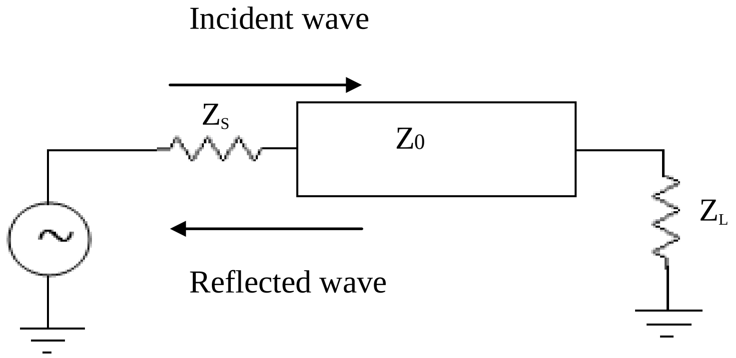


Figure 4.1 Incident and reflected waves on a transmission line.

As shown in Figure 4.1, voltage, current or power emanating from a source impedance (Z_S) and delivered to a load (Z_l) can be considered to be in the form of incident and reflected waves traveling in opposite directions along a transmission line of characteristic impedance (Zo). Cases change according to how the transmission line is terminated. So we will consider about these cases to understand the relationship between the S-Parameters and the transmission line.

When the transmission line is terminated in its characteristic impedance it means that Z_l is equal to Z_o. In this case, maximum power is transferred to the load and there is no reflected wave because of incident wave is totally absorbed in the load. Due to having no reflections, energy flows in only one direction. So it is seen that the envelope of the RF signal versus distance along the transmission line shows no standing waves.

When the transmission line is terminated in a short circuit (which can sustain no voltage and therefore dissipates zero power), a reflected wave is launched back along the line toward the source. The reflected voltage wave must be equal in magnitude to the incident voltage wave and be 180 degrees out of phase with it at the plane of the load. The reflected and incident waves are equal in magnitude but traveling in the opposite directions.

If the transmission line is terminated in an open-circuit condition (which can sustain no current), the reflected current wave will be 180 degrees out of phase with the incident current wave, while the reflected voltage wave will be in phase with the incident voltage wave at the plane of the load. This guarantees that the current at the open will be zero. The reflected and incident current waves are equal in magnitude, but traveling in the opposite directions. For both the short and open cases, a standing wave pattern is set up on the transmission line. The voltage valleys will be zero and the voltage peaks will be twice the incident voltage level.

If the transmission line is terminated with say an ideal 25 Ω resistor, resulting in a condition between full absorption and full reflection, part of the incident power is absorbed and part is reflected. The amplitude of the reflected voltage wave will be one-third that of the incident wave, and the two waves will be 180 degrees out of phase at the plane of the load. The valleys of the standing-wave pattern will no longer be zero, and the peaks will be less than those of the short and open cases. (Agilent Technologies, 2012)

When the transmission line is not terminated in its characteristic impedance, it means that Z_l differs from Zo, some of the incident wave signal is not absorbed by the load and it is reflected back to the source. The equality or inequality between the Zs and Zo determines whether the reflected wave will occur or not. If Zs is equal to Zo, the reflected wave from the load is absorbed by the source and no further reflections will occur. In the case of Zs not equal to Zo a portion of the reflected wave from the load is re-reflected from the source back toward the load and the entire process repeats itself continuously for a dissipationless transmission line. The degree of mismatch between Zo and Z_l or Zs, determines the amount of incident wave that is reflected. The ratio of the reflected wave to the incident wave is known as the reflection coefficient and is simply a measure of the quality of the match between the transmission line and the terminating impedances. The reflection coefficient, shown as Γ, is a complex quantity expressed as a magnitude and an angle in a polar form.

Γ = V reflected/V incident

$\Gamma = \rho \angle \theta$ (Eq.1)

As the match between the characteristic impedance of the transmission line and the terminating impedances improves, the reflected wave becomes smaller. Therefore using Eq. 1, the reflection coefficient decreases. When a perfect match exists, there is no reflected wave and the reflection coefficient is zero. If the load impedance is an open or short circuit none of the incident power can be absorbed in the load and all of it must be reflected back toward the source. In this case, the reflection coefficient is equal to one or a perfect mismatch. Thus, the normal range of values for the magnitude of the reflection coefficient is between zero and one. The reason normal is stressed is that in order for the reflection coefficient to be greater than one, the magnitude of the reflected wave from a load impedance must be greater than the magnitude of the incident wave to that load. In order for that to occur, it follows that the load must be a source of power…. Reflection coefficients can be plotted directly on the Smith Chart and the corresponding load impedance read off of the chart without a need of any calculations…. Given a specific characteristic impedance of a transmission line and a load impedance, the reflection coefficient can be read directly from the chart. No calculation is necessary…. Let us now insert a two-port network between the source and the load in the circuit Fig. 3.2. The following may be said for any travelling wave that originates at the source:

1. A portion of the wave from a source and incident wave upon the two port device (a_1) will be reflected (b_1) and another portion will be transmitted through the two-port device.
2. A fraction of the transmitted signal is then reflected from the load and becomes incident upon the output of the two-port device (a_2).
3. A portion of a signal (a_2) is then reflected from the output port back toward the load (b_2), while a fraction is transmitted through the two-port device back to the source.

Concluding the above, any travelling wave is made up of two components. For instance, the total travelling wave component flowing from the output of the two-port device to the load is actually made up of that portion of a_2 which is reflected from the output of the two-port device, plus that portion of a1 that is transmitted through the two-port device. Similarly, the total wave flowing from the input of the two-port device toward the source consists of the portion that is reflected from the input port (a_1) and the fraction of (a_2) that is transmitted through the two-port device. (Bowick, C., Blyler, J., Ajluni, C. , 2007)

The transmission coefficient is defined as the transmitted voltage divided by the incident voltage. For having knowledge about the system gain or dissipation we should consider about the absolute values of incident and transmitted voltages. If the absolute value of the transmitted voltage is greater than the absolute value of the incident voltage, it is said that the system has gain. If the absolute value of the transmitted voltage is less than the absolute value of the incident voltage, the system is said to have attenuation or insertion loss.

Beside these observations, the wave functions used to define S-Parameters for a two port network can be represented as below:

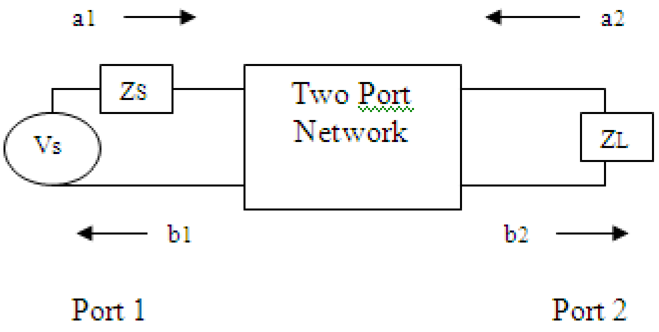


Figure 4.2 An example of two port network.

In the circuit a_1 and a_2 are defined as incident waves, b1 and b2 are defined as reflected waves.

$b_1=S_{11}\ a_1 + S_{12}\ a_2$

$b_2=S_{21}\ a_1 + S_{22}\ a_2$

where

S_{11} is the input (port-1) reflection coefficient: $S_{11} = b_1/a_1;\ a_2 = 0$

S_{22} is the output (port-2) reflection coefficient: $S_{22} = b_2/a_2;\ a_1 = 0$

S_{21} is the forward transmission coefficient (forward gain): $S_{21} = b_2/a_1;\ a_2 = 0$

S_{12} is the reverse transmission coefficient (reverse gain or loss): $S_{12} = b_1/a_2;\ a_1 = 0$

S_{11} and S_{21} are determined by measuring the magnitude and phase of the incident, reflected and transmitted signals when the output is terminated in a perfect Zo load. This condition guarantees that a_2 is zero. S_{11} is equivalent to the input complex reflection coefficient or impedance of the network, and S_{21} is the forward complex transmission coefficient. (Yarman, B.S., 2010)

Likewise, by placing the source at port 2 and terminating port 1 in a perfect load (making a_1 zero), S_{22} and S_{12} measurements can be made. S_{22} is equivalent to the output complex reflection coefficient or output impedance of the network, and S_{12} is the reverse complex transmission coefficient. (Sischka, F. 2002)

The relationship between reflection coefficient and impedance is the basis of the Smith Chart transmission-line calculator. The reflection coefficients S_{11} and S_{22} can be plotted on Smith charts and can be converted directly to impedance. By plotting S_{11}, input impedance of the two-port device can be found and similarly by plotting S_{22}, output impedance can be found easily. So that impedance can be found easily and provides optimizing a circuit design. S-Parameters are linear by default so they represent the linear behavior of the two-port. It is important to set a_1 and a_2 to zero for measuring the individual S-Parameters. It is possible to do this by terminating a network (source and load) or forcing Z_s and Z_l to be equal to the characteristic impedance. Hence, all reflections from the termination can be eliminated.

As it is mentioned at the beginning of this chapter, S-Parameters are preferred in microwave design because of being simple and easier to measure at high frequencies rather than other kinds of parameters like H, Y and Z parameters. All the given parameters provide an advantage to predict circuit performance under any source and load. Transfer and impedance matrices are all commonly used at lower frequencies and give linear behavioral model of the device and measure some factors versus frequency under different source and load conditions. But it becomes difficult to use them while operating at high frequencies, so S-Parameters are preferred. For example Y-parameters use input and output voltages and currents to describe the operation of a two-port network. Conversely scattering parameters use normalized incident and reflected traveling waves in each network port, so they are more useful design than the others. Y and Z parameters shall be measured under open and short circuit conditions, but this situation sometimes causes instability in the circuit. In spite of meeting with the instability in Y and Z parameters, there is very little risk like this while measuring with S-Parameters. Because terminating the circuit with open and short circuit conditions is not a measuring method in S-Parameters.

Instead of using open and short circuit conditions in S-Parameters, measurements are performed under natural 50 ohm terminated condition for high frequencies. Thus, effect of causing active devices to oscillate or self destruct is disappeared. S-Parameters are defined with respect to reference impedance. The network is usually terminated in the characteristic impedance of the measuring system that is typically 50 ohms (purely resistive). It is also possible to compute other two-port parameter sets such as H, Y, and Z parameters from S-Parameters.

Scattering parameters which are referred to as S-Parameters, completely and uniquely define the small signal gain. S-Parameters are a parameter set that relates to the traveling waves that are scattered or reflected when an n-port network is inserted into a transmission line. The degree of scattering is an important term with regards to S-Parameters. Because it provides understanding all linear characteristics of the device based on the degree of scattering. Scattering in this sentence means reflection back to the source and transmission to other directions.

The transfer of signals or power can be expressed by S-Parameters. These parameters can easily compute the physical quantities such as attenuation of a filter, stability of a transistor, voltage gains or power gain of the circuit. (Agilent Technologies, 2012)

CHAPTER FIVE
CIRCUIT SIMULATION USING ADS SOFTWARE

When ADS software is started, user meets two separate parts in Getting Started with ADS window. One part is Projects, the other is HelpCenter. It is possible to learn how to use ADS software from Help Center. On the left part of the menu there are the methods of creating projects and opening an existing project or an example project. It is shown in Figure 5.1.

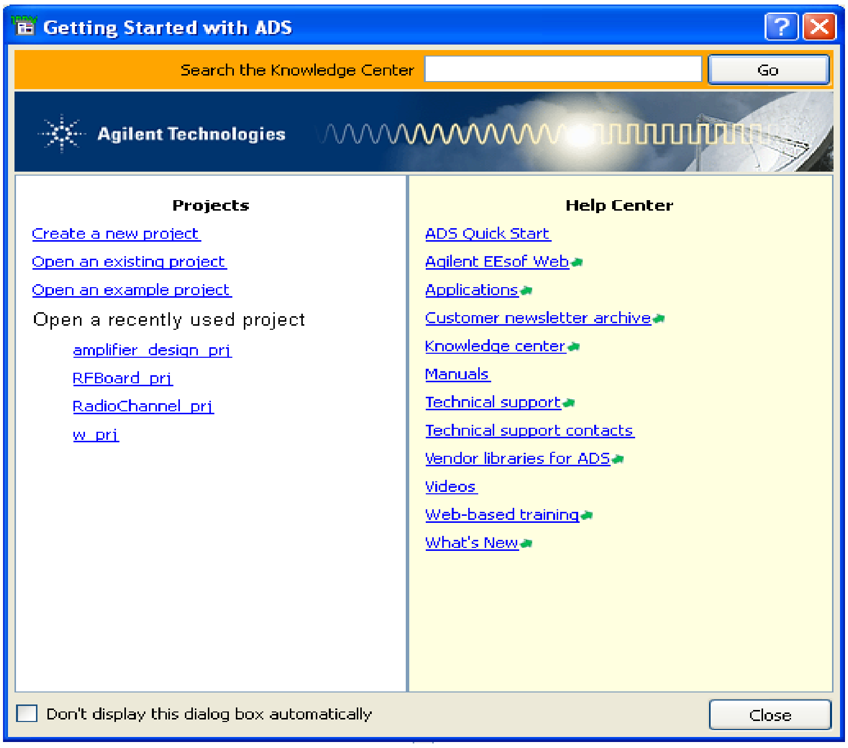


Figure 5.1 Getting started with ADS window.

ADS Program uses projects to organize and store the data generated when you create, simulate, and analyze designs to accomplish your design goals. A project includes circuit, layout, simulation, analysis, and output information on the designs that you create, along with any links you add to other designs and projects. (Yarman, B.S.,Şengül,M.,2008)

Also the Advanced Design System Main window can be used to create and open projects besides the Getting Started with ADS menu. This window is also called main window. It is shown in Figure 5.2.

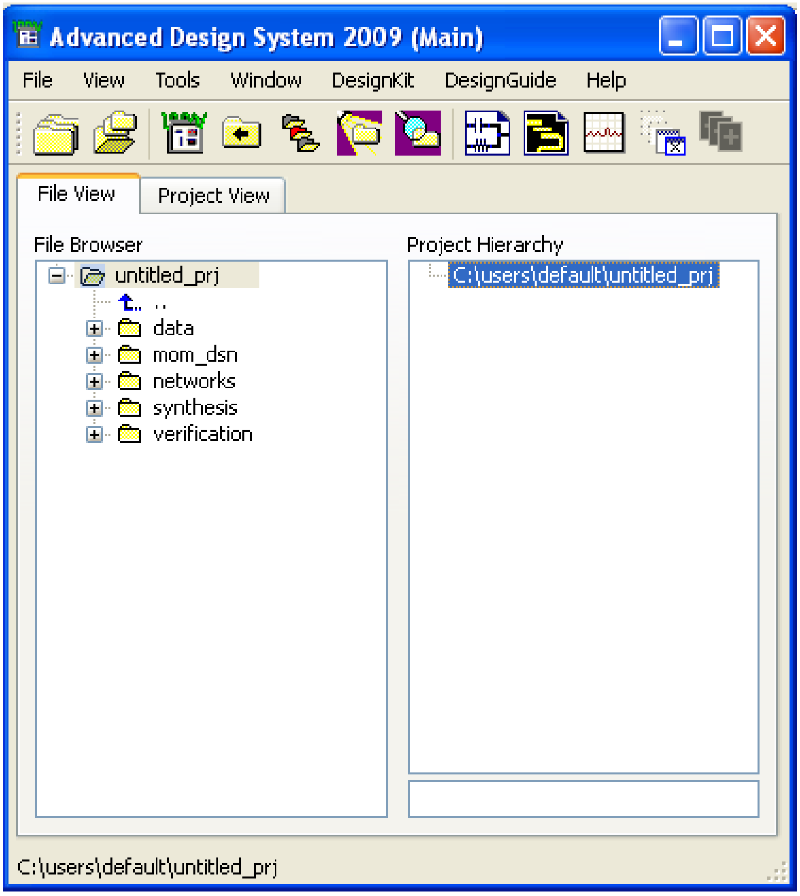


Figure5.2ADS main window is used to create and open projects.

Circuit schematics (networks), and simulation results (data) are organised into a project. Each project is self contained piece of work. Physically a project is a directory which contains several other directories. Your schematics are stored in a directory called “Networks”. Each schematic is stored in a separate file with the extension “dsn”. The results from your simulations are stored in the directory data with the extension “ds”. The other directories and files store information required for ADS to correctly work and should not be deleted or edited. By default ADS adds “PRJ” to the end of the project name.(Yarman, B.S., Şengül, M.,2008)

5.1 Creating Project

A new project is created by selecting “file > New Project” from the Main window. When New Project window is appeared by pressing on it, a dialog box is opened. This dialog box allows user to browse a Project or give a new name and to select the length unit of the circuit. Millimeter is an advisable choice for the length unit. It is shown in Figure 5.3.

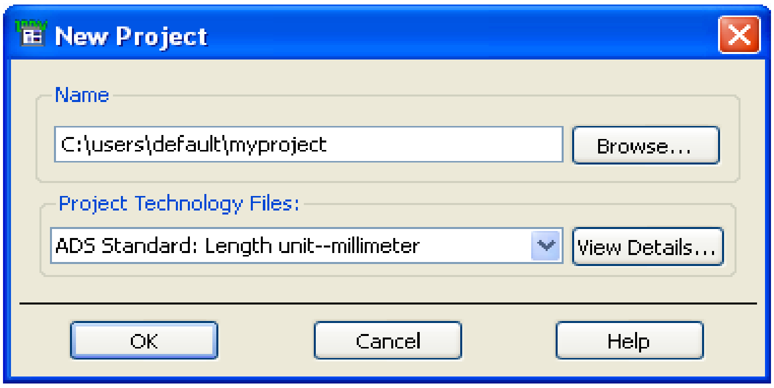


Figure 5.3 New project window is used to browse a project or give a newname and to select the length unit of the circuit.

5.2 Creating Design and Performing Schematic

After adjusting the parameters related to creating project, a schematic wizard is opened. There are many options on this schematic wizard, but it is not necessary. It is advisable to select “No help needed” or to cancel the wizard. Schematic Wizard is shown in Figure 5.4.

After closing Schematic Wizard, Schematic window with the project name given by user is appeared. On the top of window there are many short cuts of commands. Some of the short cuts are insert port, insert ground, simulate, insert wire, end command etc. These buttons provide quick access. On the left of menu there are set of components which is used to create circuits by drag and drop. There are many choices like lumped-components, sources, simulation types, devices, filters and etc while selecting components. If mouse is moved on a palette index or icons on the top of schematic, then pop up windows are appeared. These windows help user to have information about that icon. It is shown in Figure 5.5.

Figure 5.4 Schematic wizard window.

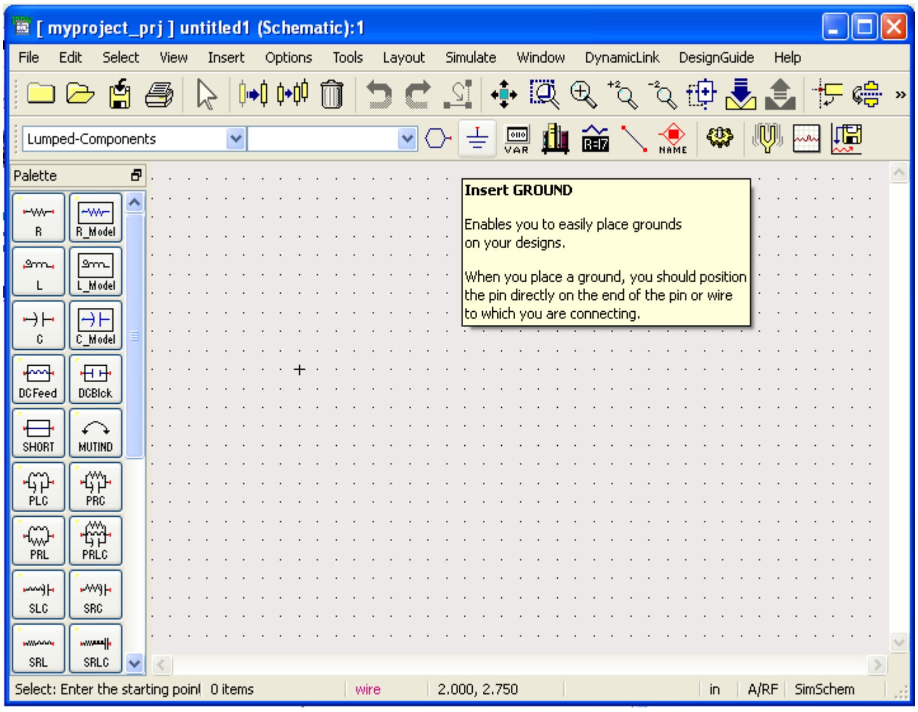


Figure 5.5 It is possible to create many circuits via the schematic window.

For studying on an example project, "open an example project" should be selected from the menu shown in Figure 5.1. It is also possible to open a Project via "file > Open Project". There are circuit, layout, simulation, analysis and output information in a Project. Besides selecting components from the left side of menu, a specific component can also be found from "view > component > component library." After selecting component from library or dragging and dropping it from the left side, some adjustments should be done on the component.

By double clicking on the component, parameter box is opened and component value and other options are entered. This information can be seen next to the component after adjusting and closing the parameter box. A parameter box of capacitor is shown in Figure 5.6 as an example.

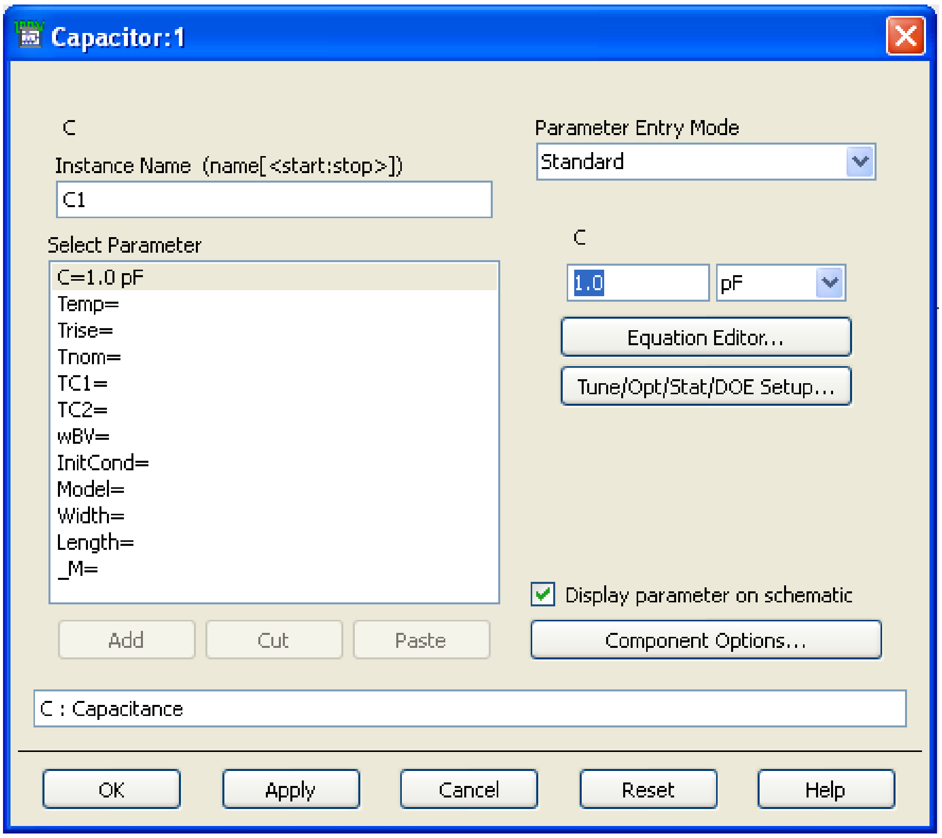


Figure 5.6 Parameter box is used to adjust the parameters of circuit element- e.g. capacitor.

After choosing the components which is used to create a circuit, they should wire up with each other. To connect the components "Insert wire" button is used. "Insert port" is used to place ports and "Insert ground" is used to place ground on designs.

5.3 Simulating the Design

When the circuit is completely created, a simulation controller should be located on the circuit. The simulation type should been selected from the left side of menu. "Simulation" box is selected from the top of menu to simulate the design.

After pressing the "Simulation" box, simulation is done and a new window is opened referred to as "Data Display Window". In this window there is a palette to draw the simulation results of designed circuit. When choosing a plot type from the palette, it is enough to place it on data display window by drag and drop.

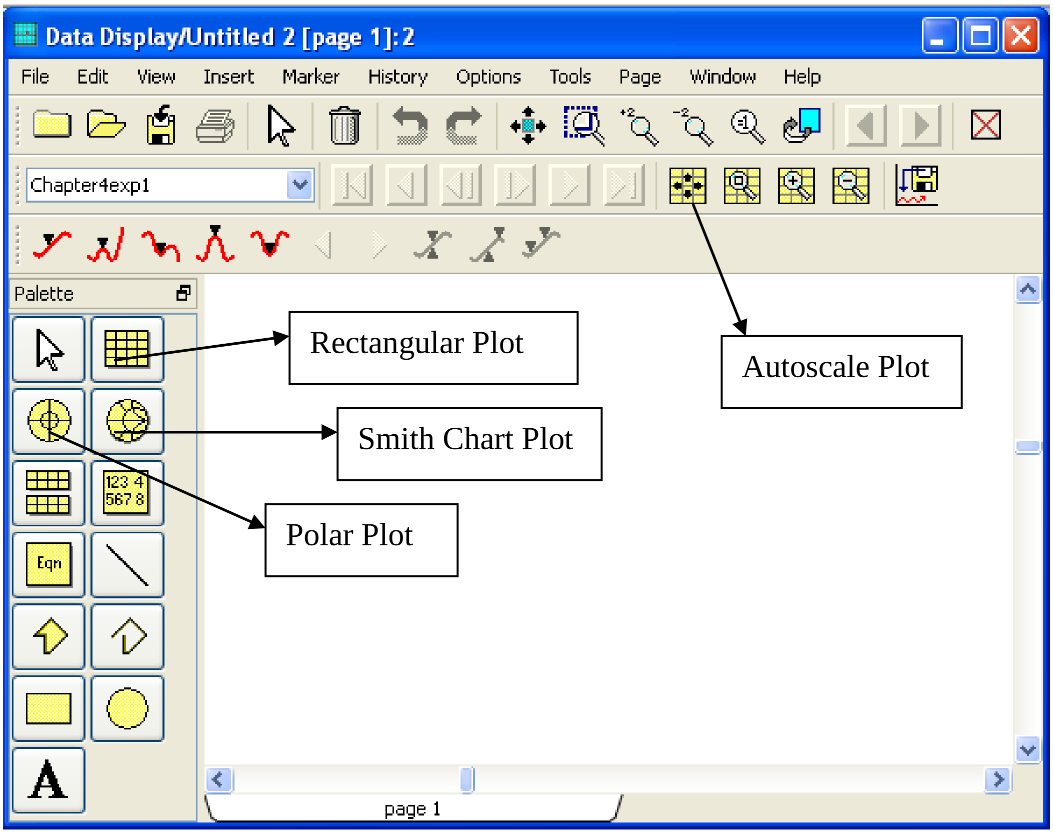


Figure 5.7 Data display window allows displaying results of simulation.

After placing the plot, another new window is opened called as "Plot Traces & Attributes". This dialog box allows changing settings of the plots and choosing the parameters which we want to draw. This window can be seen in Figure 5.8. After selecting the simulation graph, simulation result is shown in data display window.

On the same window some qualifications such as magnitude, real part or imaginary part of the parameters can be chosen. From the trace options menu, lots of function can be set such as color, thickness and etc.

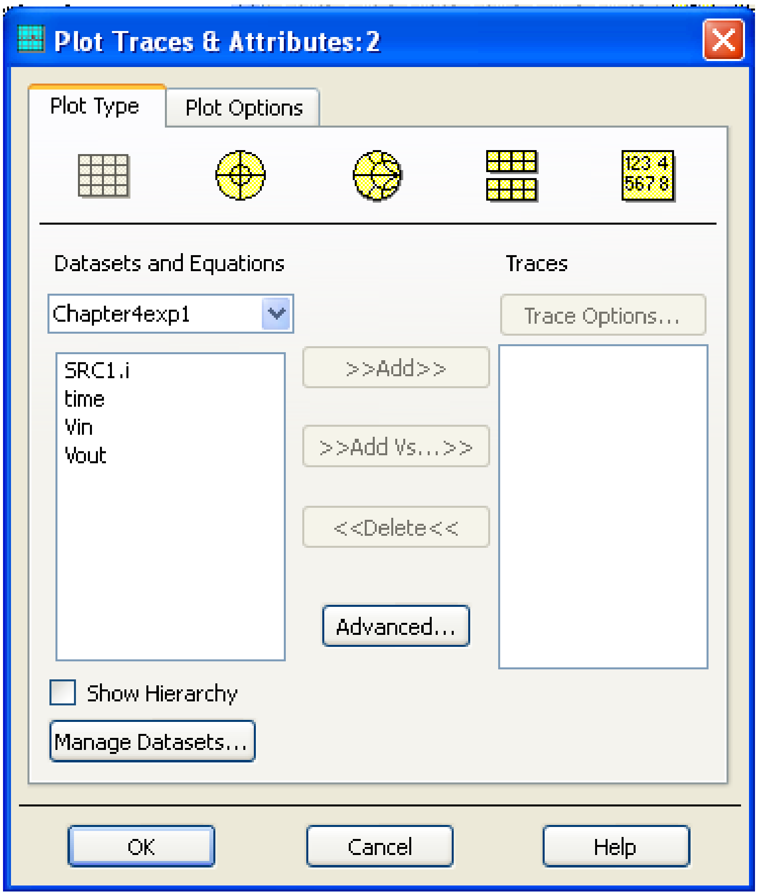


Figure 5.8 It is possible to change plot setting from this window.

5.4 Performing Examples in ADS Software

In the light of all this information described above, performing examples of designing and simulating circuits help to increase understandability of ADS software.

5.4.1 An Example of RC Circuit

Let us perform a simple RC circuit and simulate it. The components which are used in this example circuit can be seen in Figure 5.10. Circuit components have been chosen from lumped-components menu. Settings of components have been done via parameter box. After placing the components and setting the values of them, “Pulse” component has been selected from the “Source-Time Domain” menu. “Transient Simulation” has been chosen as a simulation type from the “Simulation” menu. The wire between the source and resistor is named as Vin and the wire between the resistor and capacitor is named as Vout. This adjustment is done via Wire/Pin Label window shown in Figure 5.9.

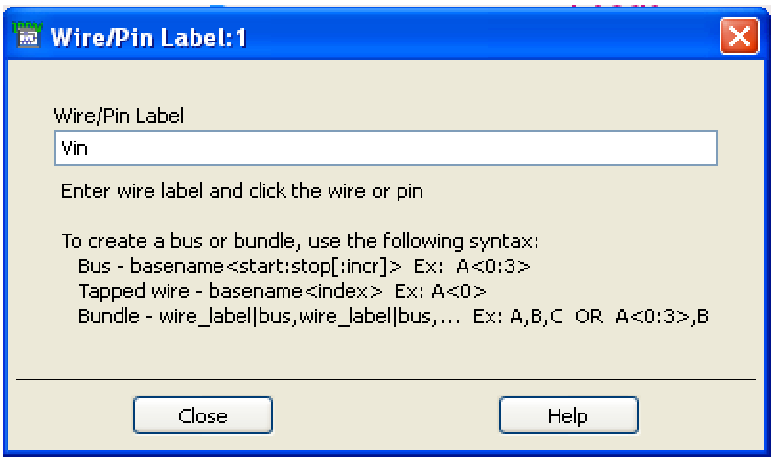


Figure 5.9 An example of a simple RC circuit.

After saving the design, next step is clicking on the simulate icon.

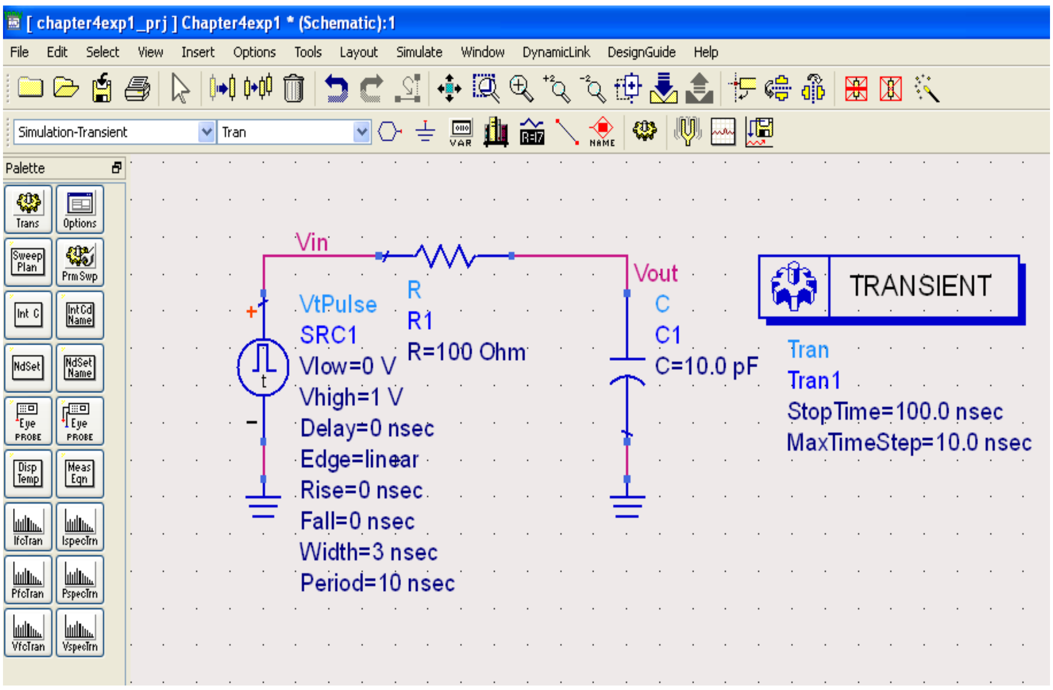


Figure 5.10 An example of a simple RC circuit.

After the simulation has been completed, data window has been appeared. The data of Vin and Vout have been added to traces menu and in rectangular plot form. Simulation result is shown in Figure 5.11.

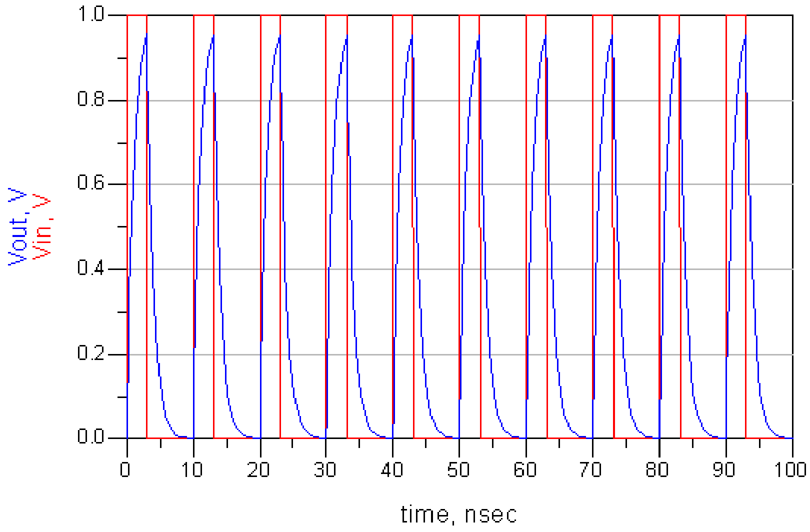


Figure 5.11 The transient response of RC circuit.

5.4.2 Performing an Example of Class A Amplifier Circuit

A Class A amplifier has been designed as another circuit example. While designing the circuit, current probe called as “I_Probe” has been chosen from “Probe Components” menu. Using this item provides to see the current value flowing from the base of transistor.A transistor called BIP1 type has been used with various resistors, capacitors and inductances. A view of circuit schematic is shown in Figure 5.12. After simulating the circuit, rectangular plot has been chosen from data display window. To obtain the graph of the transfer function of the circuit, dB(Vout/Vin)and phase(Vout/Vin) data have been added to trace menu versus log frequency axis. Hence, it is possible to see the frequency response of the circuit. Bode plot is a combination of Bode magnitude plot and Bode phase plot. Circuit simulation results can be seen in Figure 5.13 and Figure 5.14.

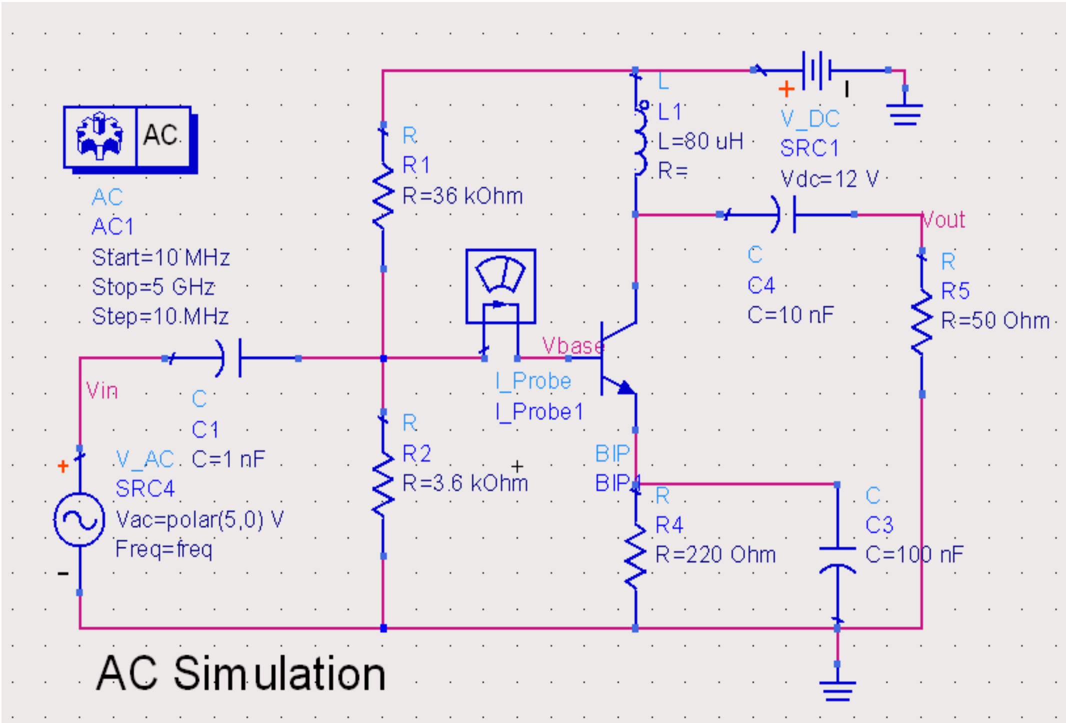


Figure 5.12 An example of class a amplifier.

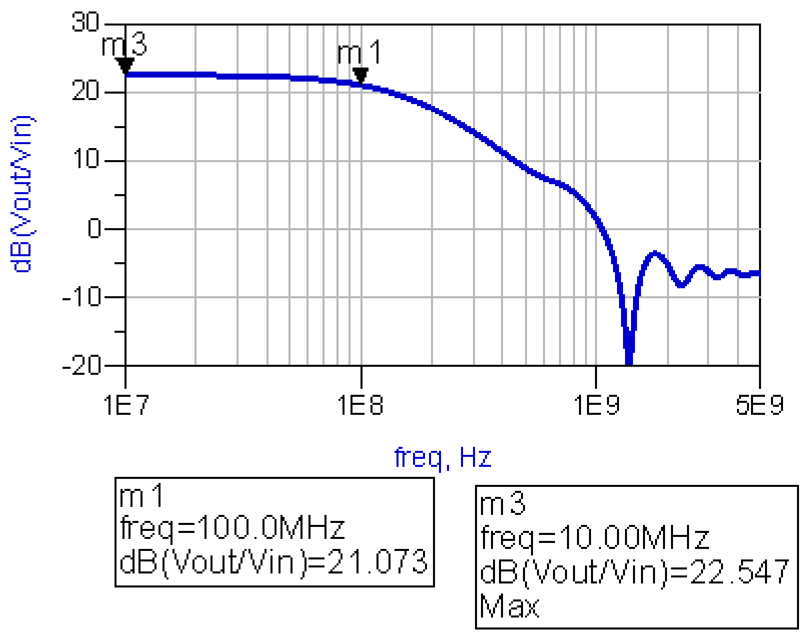


Figure 5.13 Bode magnitude plot of the amplifier.

It is also possible to see the phase(Vout/Vin) versus log frequency axis.

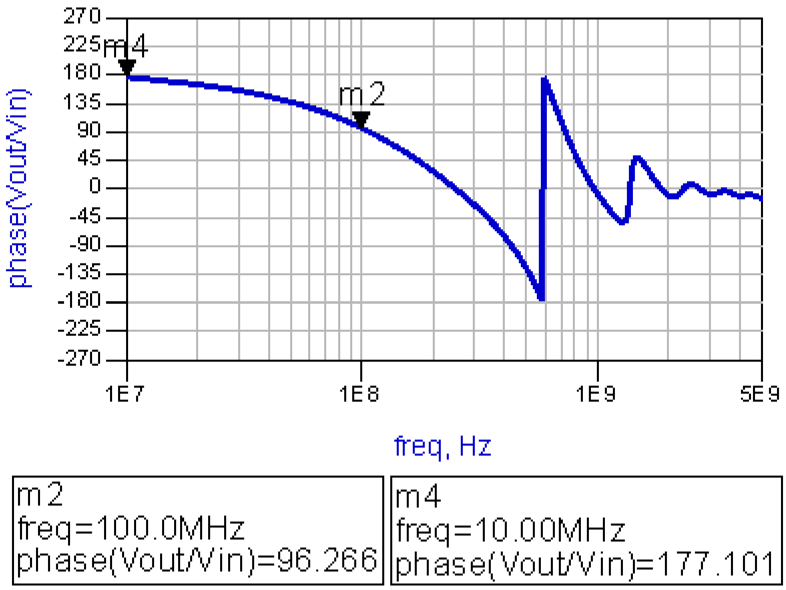


Figure 5.14 Bode phase plot of the amplifier.

5.4.3DesigningExamples of Low Noise Amplifier Circuit

A low noise amplifier has been designed as another circuit example. Primary purpose of a low noise amplifier is to amplify the input signal while keeping noise as little as possible. A transistor called HBFP-0450 type has been used with various resistors, capacitors and inductances. S-Parameters simulation has been chosen as simulation type. All the S-Parameters simulation components such as “Sweep Plan” and port impedance terminations called “Term” can be chosen from “Simulation-S_Param” menu. A view of circuit schematic is shown in Figure 5.15. While designing a low noise amplifier, it should be considered to have high gain with more bandwidth.

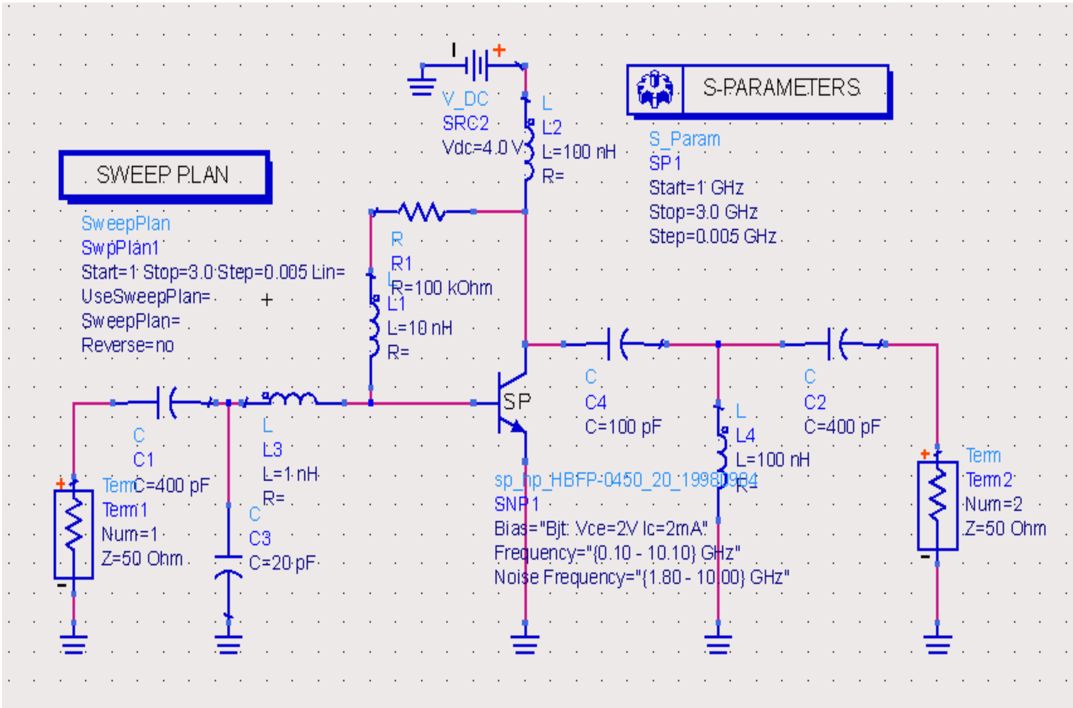


Figure 5.15 An example of low noise amplifier circuit.

After simulating the circuit, rectangular plot has been chosen from data display window. dB(S(2,1)) data called as gain has been plotted on the graph via selecting from trace menu. Frequency axis is formed in log mode. Circuit simulation result can be seen in Figure 5.16.

Marker function helps to learn the value at determined frequency. This function can be selected from the top menu among the marker options. As it is seen in this figure, maximum gain is obtained at 1565MHz with the gain value of 5.2 approximately. The decibel of input reflection coefficient called as dB(S(1,1)) has been selected from the trace menu and it is plotted versus log frequency. Circuit simulation result can be seen in Figure 5.17.

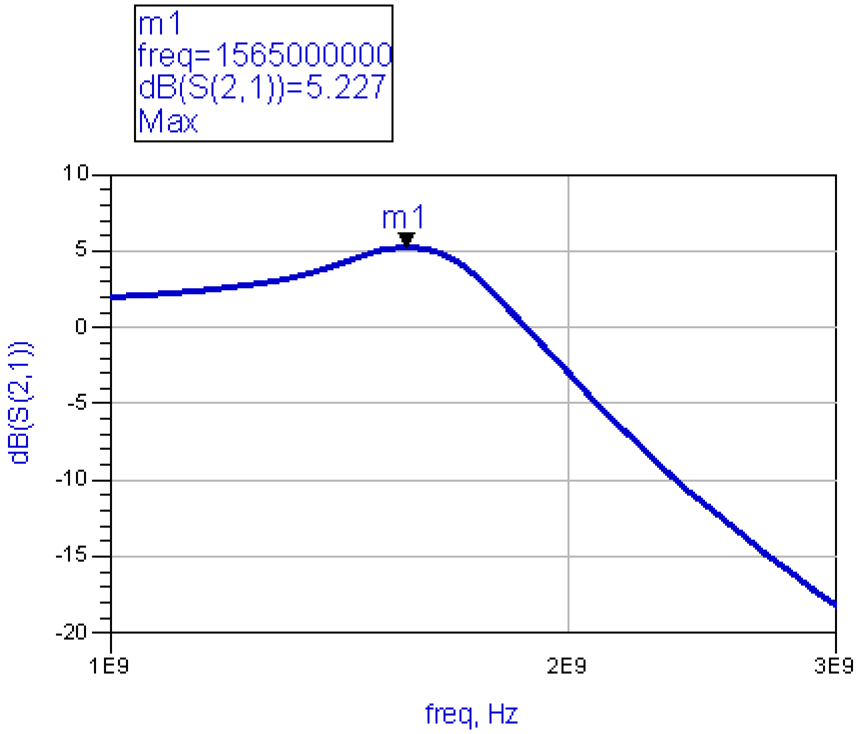


Figure 5.16 Circuit gain versus log frequency axis.

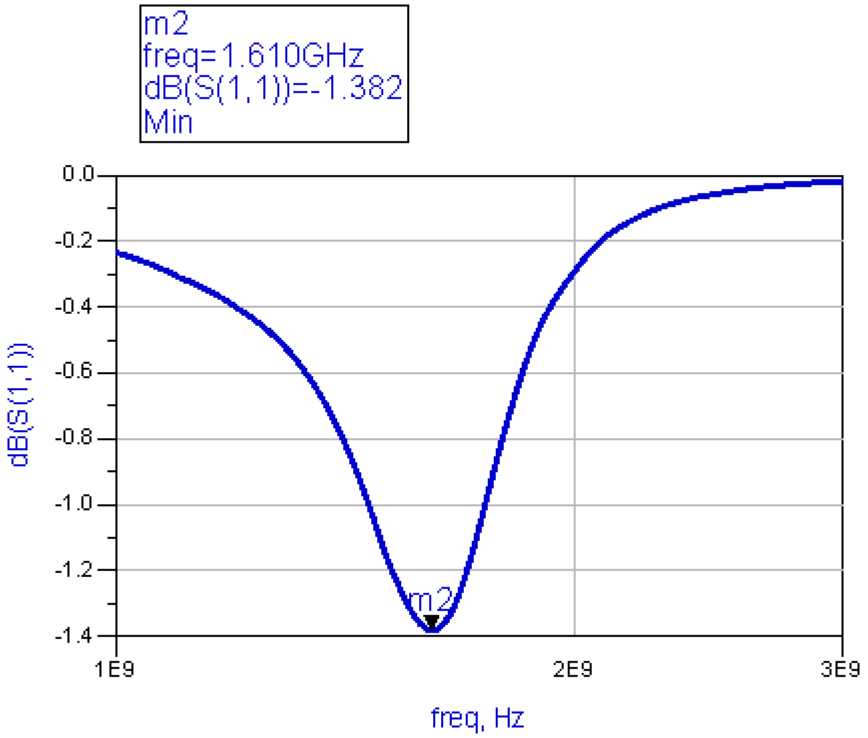


Figure 5.17 Input reflection coefficientversus log frequency axis.

In Figure 5.18, there is another example of low noise amplifier. It is a two stage amplifier. This amplifier consists oftwo transistorswith NPN type, resistors and a capacitor. Bode magnitude plot, in other words dB(Vout/Vin) versus log frequency axis can be seen in Figure 5.19.

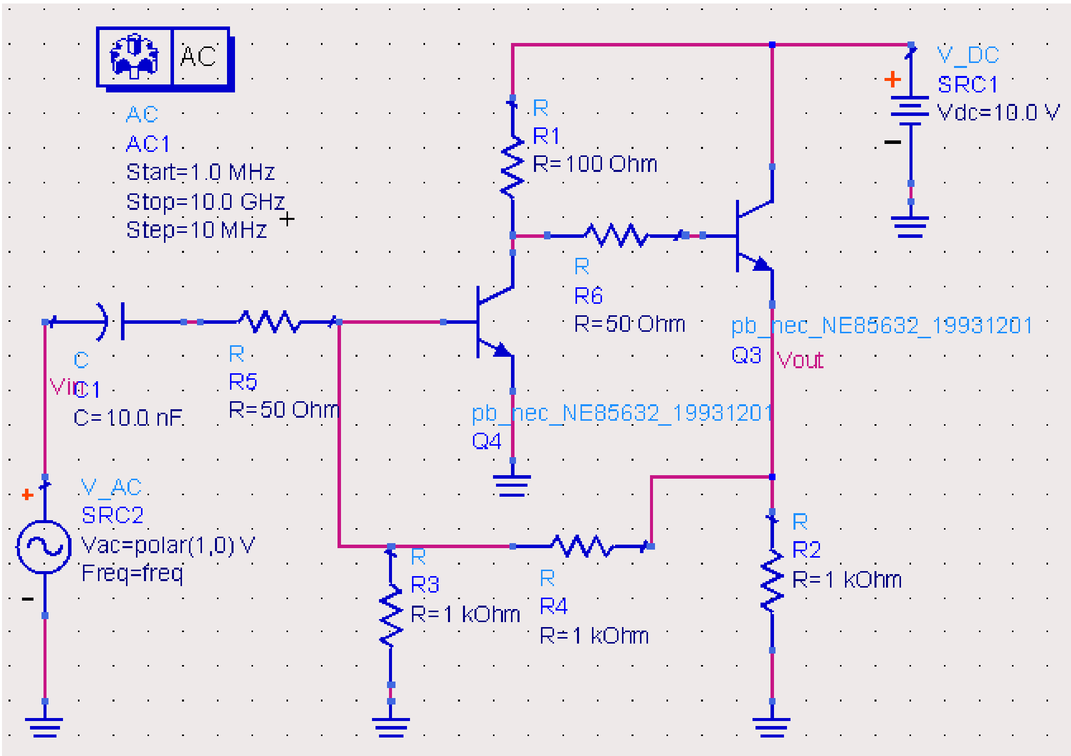


Figure 5.18 Another example of low noise amplifier circuit.

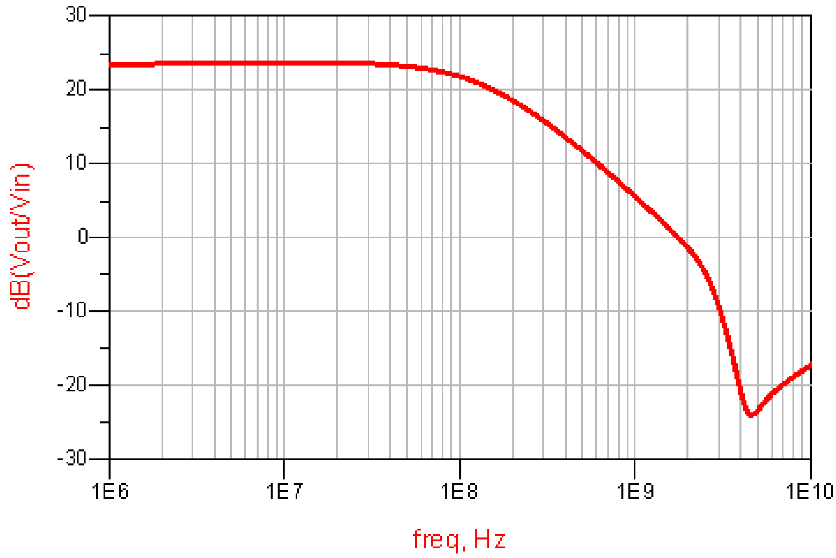


Figure 5.19 Bode magnitude plot of the low noise amplifier.

Components which are selected from the “Lumped-Components” menu do not have specified parameters.To have more accurate results, components which have specified parameters are used for example Murata components.The reasons having more accurate results with Murata components are having parameters to see and change the parasitic effects of components. Low noise amplifier design that is seen in Figure 5.15 has been reformed with Murata components to see the difference between the previous one. A view of circuit schematic with Murata components is shown in Figure 5.20.

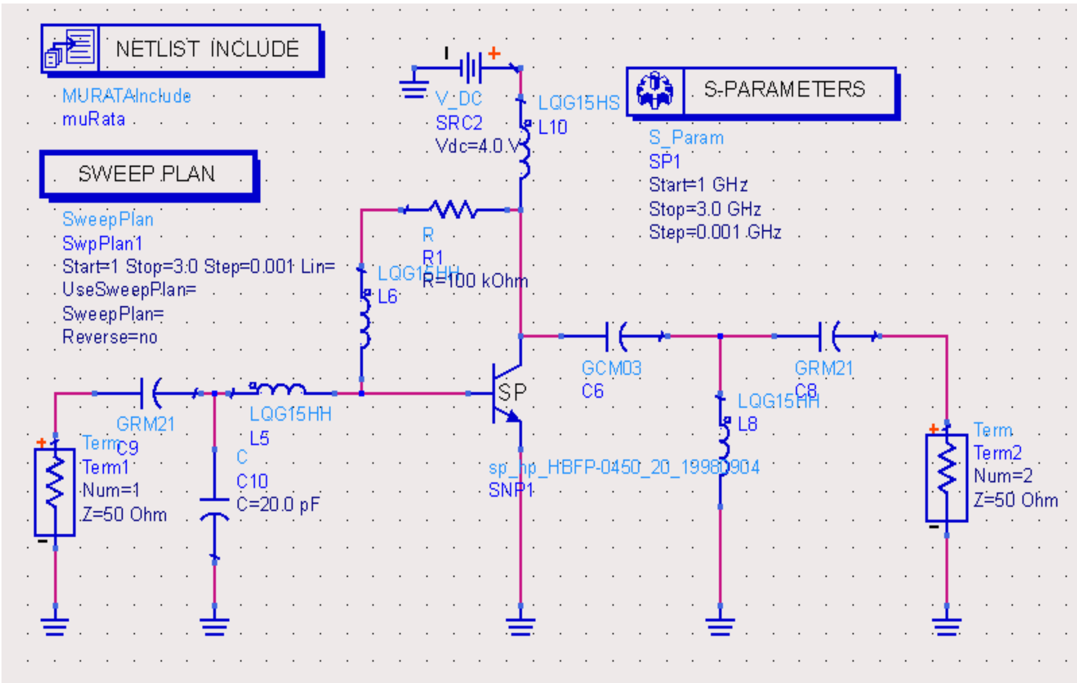


Figure 5.20An example of low noise amplifier circuit with Murata components.

Besides having the same design with Figure 5.15, the values of circuit components are the same with the values used before.

The results of the circuit are shown in Figure 5.21 and Figure 5.22. As it seen from the results it is possible to obtain more accurate results with Murata components. It is observed that the gain of the low noise amplifier and the frequency at which the maximum gain decreases in comparison with the result shown in Figure 5.16. It is possible to see the parasitic effects of components by observing the results shown in Figure 5.23.

When the Figure 5.22 is compared with the Figure 5.17, it is realized that there is a similar conclusion for the input reflection coefficient graphs.

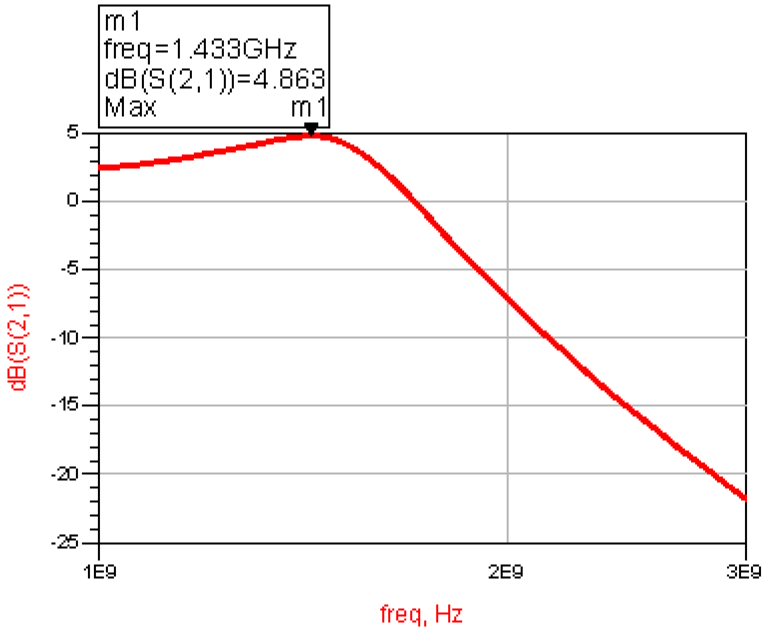


Figure 5.21 Circuit gain versus log frequency axis with Murata components.

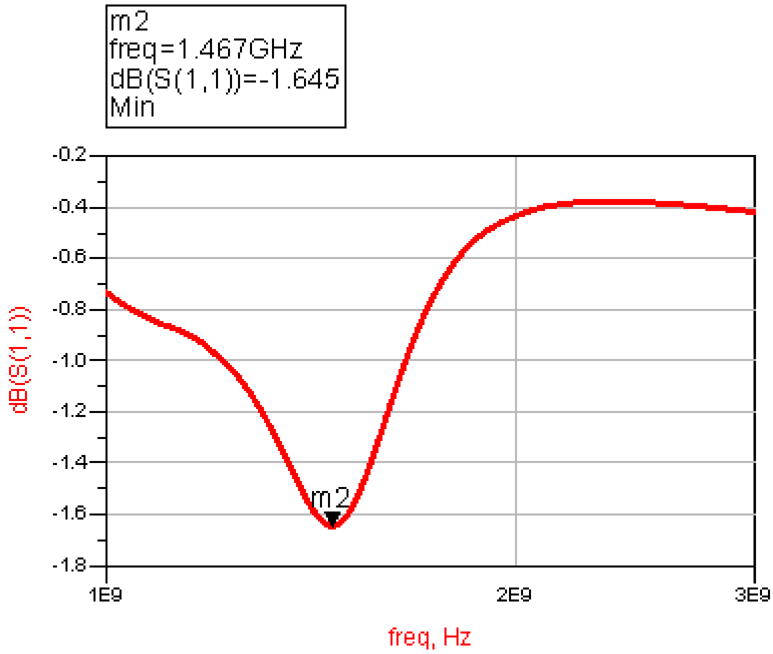


Figure 5.22 Input reflection coefficient versus log frequency axis with Murata components.

As a conclusion, it will be useful to see the response of the circuits together as top and bottom. Therefore the effect of the usage of Murata components could be seen significantly. This result can be seen in Figure 5.23.

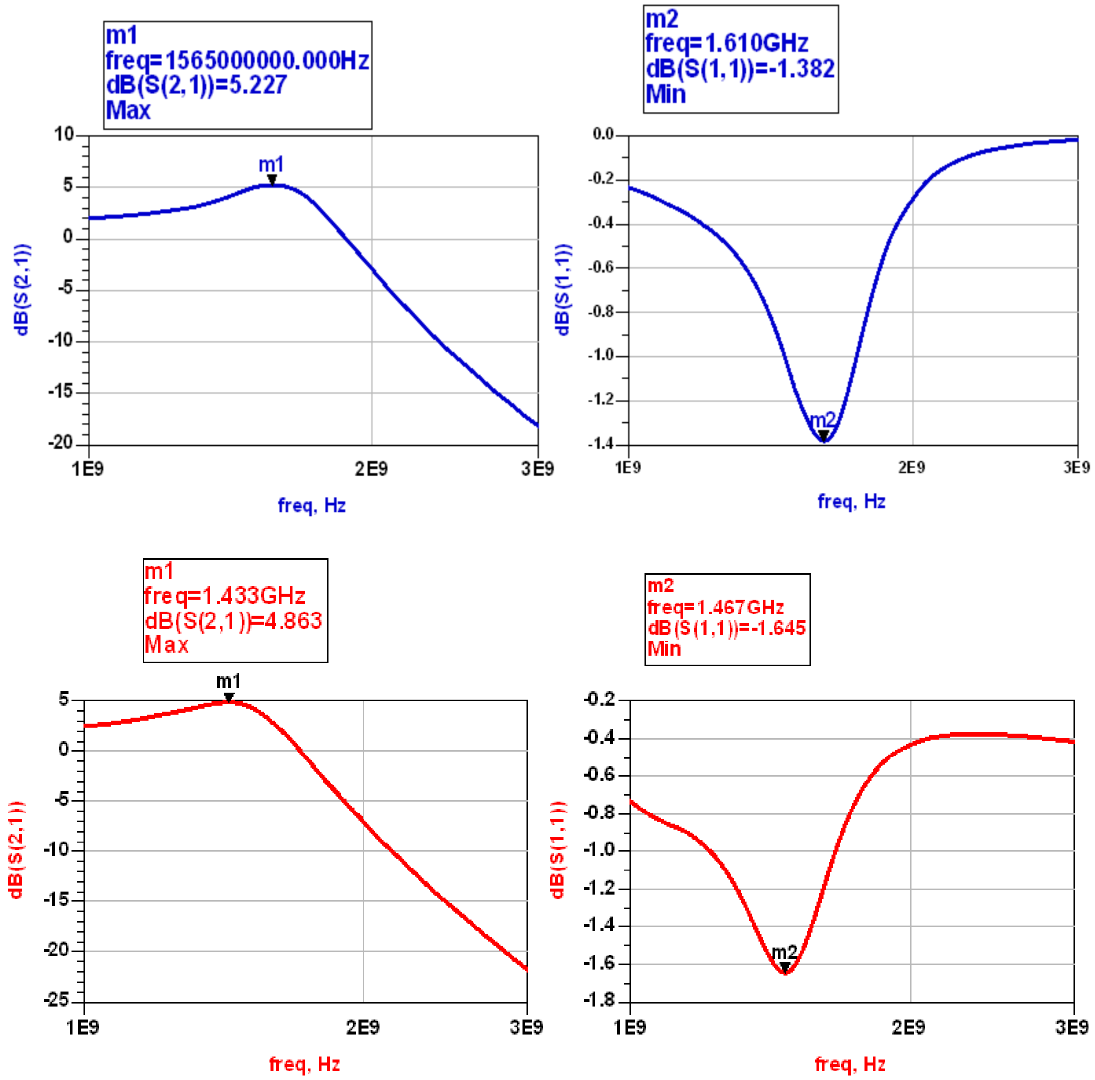


Figure 5.23Top and bottom view of the response of LNA circuits in Figure 5.15 and 5.20.

CHAPTER SIX

ELECTROMAGNETIC EFFECTS ON RF CIRCUIT DESIGN

6.1 The Theory of Microstrip Lines

“Lumped-components” menu has been used while creating the circuit in ADS so far. But it is not possible to observe the effects of transmission line, distances between the components or another effect as causing disturbances in this way. Hence, if it is required to see the response of the circuit against the electromagnetic effect, “TLines-Microstrip” menu should be used in ADS software. Thus, it can be easily observed that how the result of effects changes according to the changes on circuit. Before studying on simulating circuits in ADS, firstly it is better to have knowledge of the subject of microstrip lines.

Typical transmission line configurations are coaxial line, stripline, two-wire line, parallel plate line, co-planar line, slot line, microstrip line and shielded microstrip line. Microstrip is a preferred transmission line because of having less cost than traditional waveguide technology as well as being more compact. Microstrip is used to transmit microwave frequency. It is also possible to use it into high frequency digital circuits and MMIC (Monolithic Microwave Integrated Circuits). Microstrip lines are easy to fabricate and they are fairly suitable for coupled line structures. It is also very important to understand the operating principles of microstrip line for taking precautions about the subject of electromagnetic compatibility.

The microstrip line is a transmission-line geometry with a single conductor trace on one side of a dielectric substrate and a single ground plane on the opposite side. Since it is an open structure, microstrip line has a major fabrication advantage over stripline. It also features ease of interconnections and adjustments. (Maloratsky,Leo G.,2000)

Microstrip line consists of the dielectric and conductor parts. Conducting strip separated from ground by a dielectric layer known as the substrate. A simple designation of microstrip line is shown in Figure 5.1.

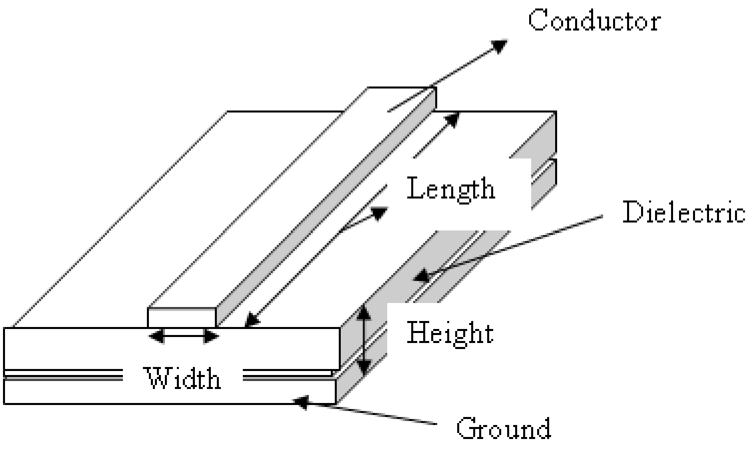


Figure 6.1 The cross-section of microstrip line.

Besides the advantages, microstrip lines have some disadvantages such as having the generally lower power handling capacity, and higher losses. Also, because of being not enclosed, it is susceptible to cross-talk and unintentional radiation. Therefore when it is used in a filter which has to be high isolation, microstrip line shall be shielded externally to protect from the environment, electromagnetic radiation and interference. The size and shape of the metal box are the important factors when operating circuits with microstrip line.

The main purposes of the housing or package are to provide mechanical strength, EM shielding and heat sinking in the case of high-power applications. Packaging must protect the circuitry from moisture, humidity, dust, salt spray, and other environmental contaminants. In order to protect the circuit, certain methods of sealing can be used: conductive epoxy, solder, gasket materials, and metallization tape. (Maloratsky,Leo G.,2000)

6.2 Designing Low Pass Filterwith Microstrip Lines in ADS Software

After having information about the theory of microstrip lines, studying on ADS software will provide to understand the subject better. There are two ways to design and simulate in ADS software. First one is using schematic simulator; the second one is using the full wave simulator which is based on Methods-of-Moments. While simulating in schematic simulator, S-Parameter equations are used.

6.2.1 Creating Low Pass Filter with Using Schematic Simulator

On the left of schematic menu there are set of components which is placed in the component palette list. If mouse is moved on a palette index or icons on the top of schematic, then pop up windows are appeared. TLines-Microstripmenu is chosen shown as Figure 6.2.

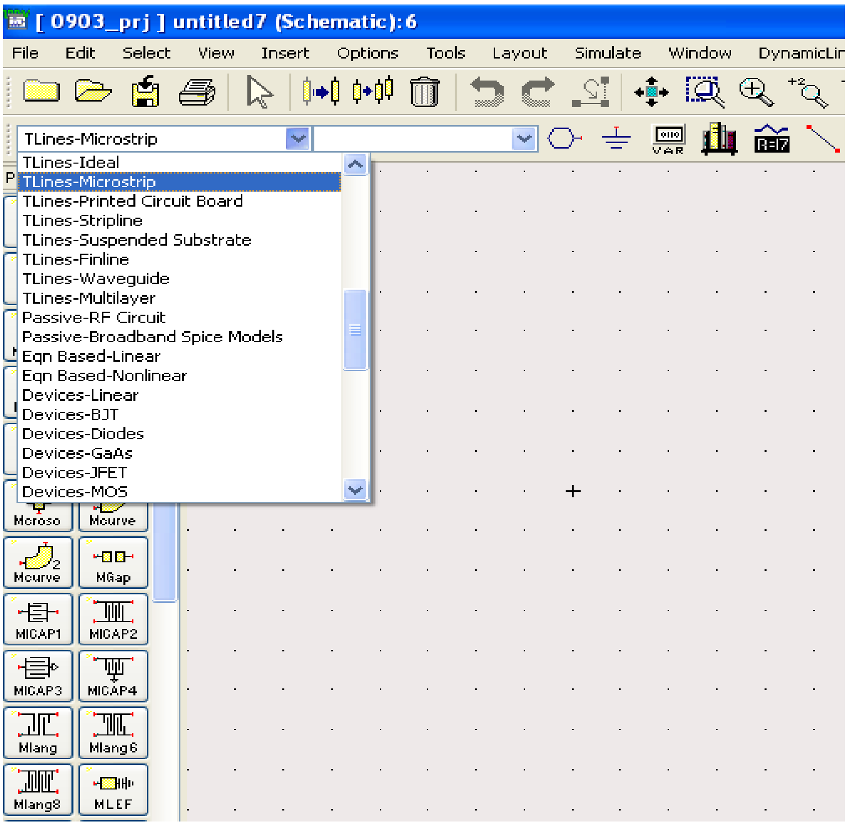


Figure 6.2 TLines-Microstrip is one of the most popular transmission structures.

MLIN component is selected for the performance of Microstrip line. After adding MLIN components, MSUB is placed on the schematic. MSUB is referred as Microstrip Substrate. It is possible to edit substrate parameters via MSUB component.

Simulation-S_Param is placed to the schematic from components palette list. Term component which is referred as port impedance termination for S-Parameters is placed on the circuit and then all the components are wired up together.

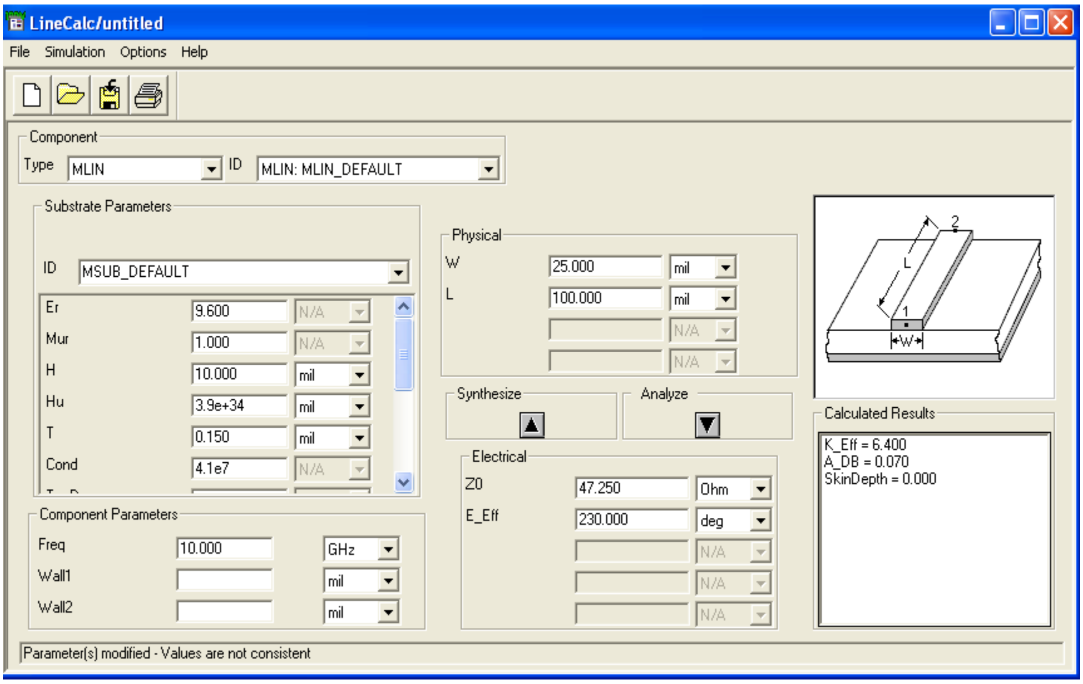


Figure 6.3 LineCalc is used to determine the physical parameters of MLIN.

MLIN components' parameters are determined via "LineCalc" menu. It is easily reached under Tools menu. In this menu besides the physical characteristics such as width, length, there are also electrical and substrate parameters.

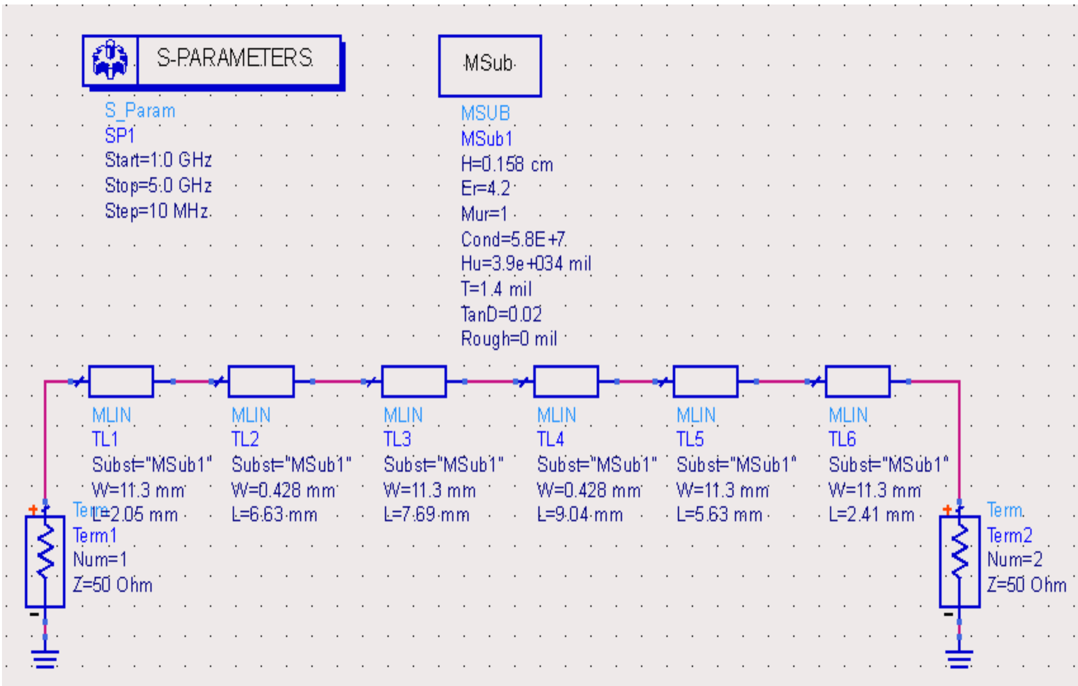


Figure 6.4 Stepped-impedance low pass filter with a cutoff frequency at 2.5GHz.

Circuit is created as it is shown in Figure 6.4. After the design of circuit is completed, it is simulated. When it is simulated, a new data display window is appeared. Rectangular plot type is selected from the palette on data display window.

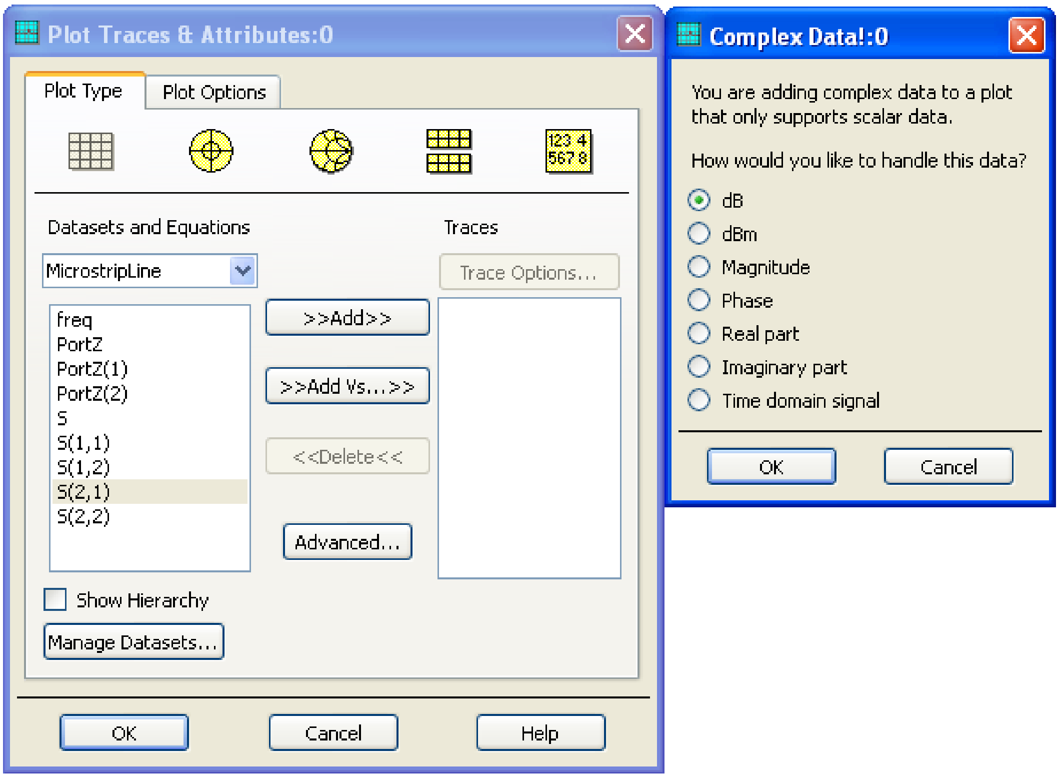


Figure 6.5 As the left window allows to change data parameters, the right one allows to specify the type ofdata.

After placing the rectangular plot, another new window is opened called as "Plot Traces & Attributes". This dialog box allows changing settings of the plots and choosing the parameters which we want to draw. S(2,1) data is selected to be plotted from this menu. When it is selected, another window is appeared as shown on the right part of Figure 6.5. "Complex Data" window is allows to specifying the type of data. dB is selected from this menu.

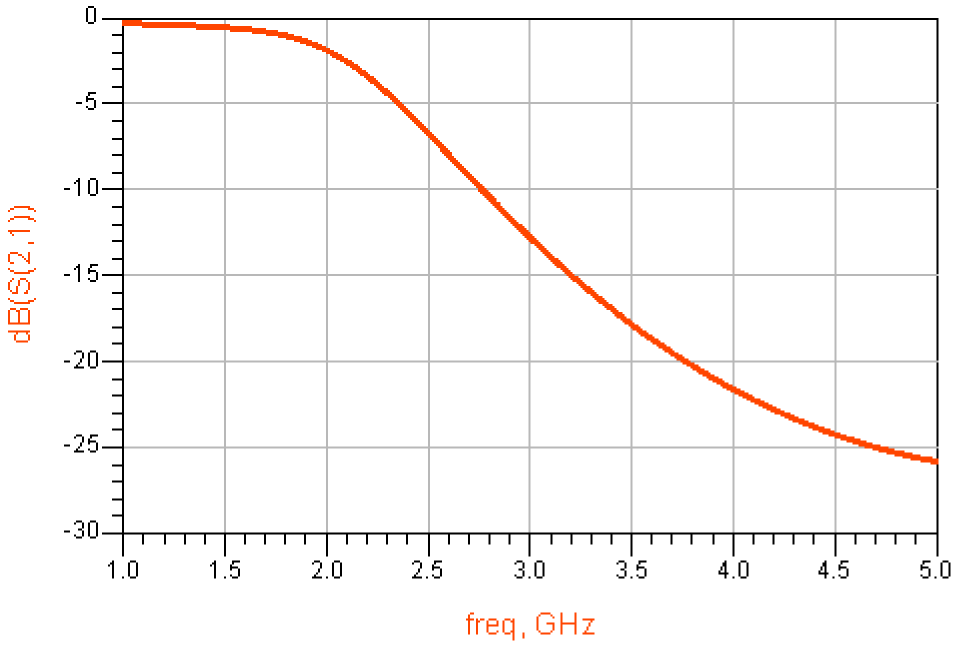


Figure 6.6 The response of low pass filter.

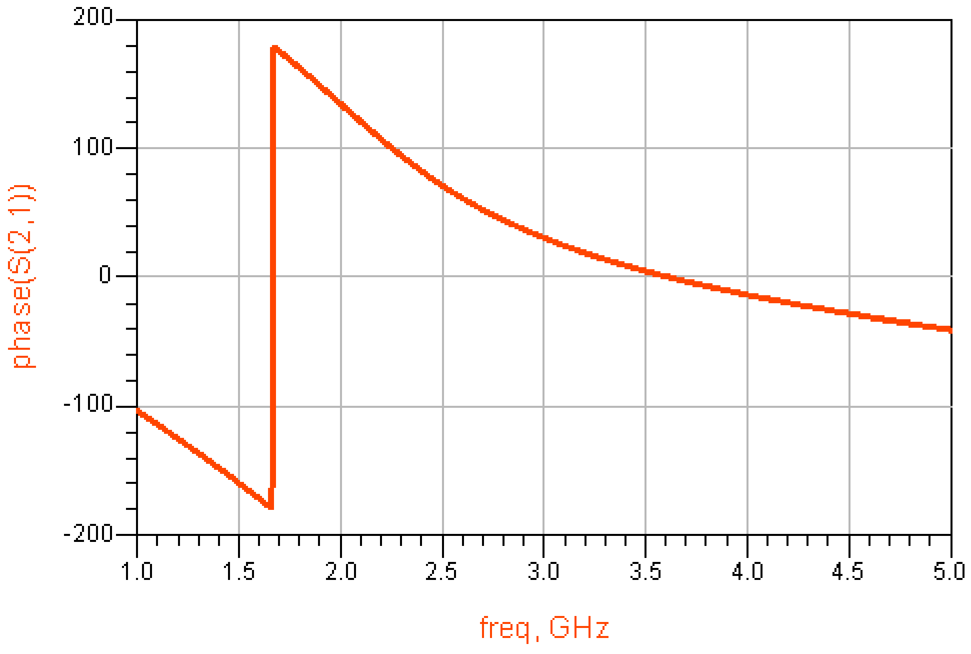


Figure 6.7 The phase (degree) versus frequency.

6.2.2 Creating Low Pass Filter with Using Full-Wave Simulator

Microstrip lines can be performed with using full-wave simulator referred as Momentum. While designing the circuit with microstrip lines via full-wave simulator "port" is inserted to the circuit instead of the "term" used in S-Parameters. As it is specified before, in circuits used with schematic simulator S-Parameter equations are used, conversely while studying on momentum Maxwell equations are used instead of S-Parameter equations.

As a result, a full-wave simulation is more accurate and should always be preformed for devices that have bends or close gaps or when the user wishes to model the device as accurately as possible. In addition, the schematic simulator does not account for radiation, therefore radiating structures such as antennas designed using Agilent ADS can only be characterized by Momentum.(EM Talk, 2011)

The second step is performing the same low pass filter example illustrated in Figure 6.4 with the full-wave simulator. The circuit design in schematic window should be saved as in another name. 50Ω terminals recalled as terms and ground symbols in the low pass filter circuit are deleted. Ports are inserted instead of terms used in S-Parameter simulation. Port icon is achieved from the top of the menu in schematic window. The new created circuit can be seen in Figure 6.8.

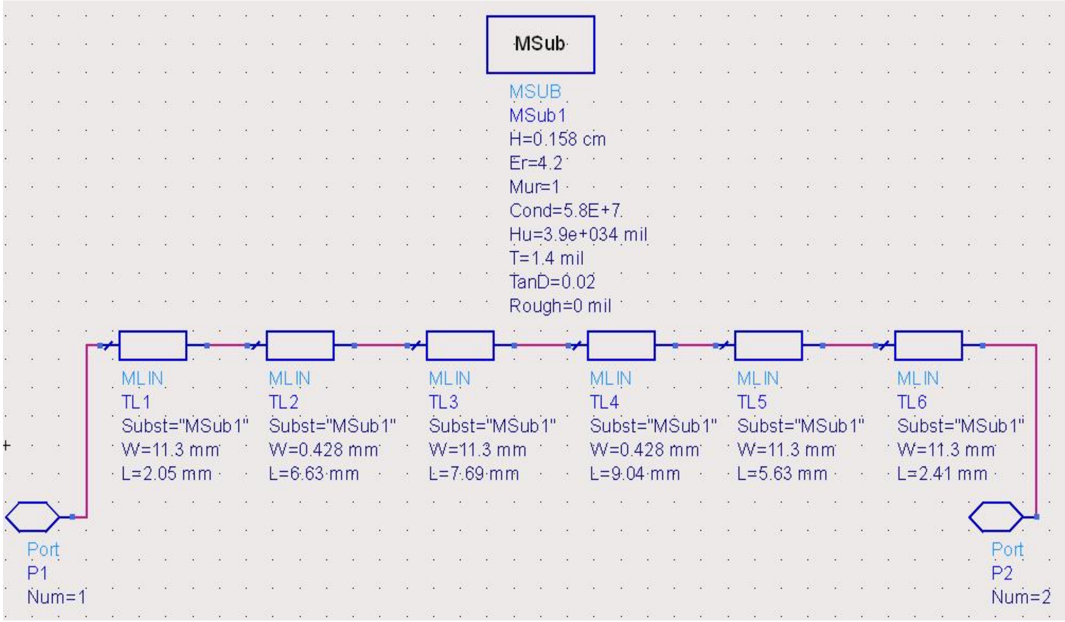


Figure 6.8 Ports are inserted to the low pass filter to simulate with full-wave simulator.

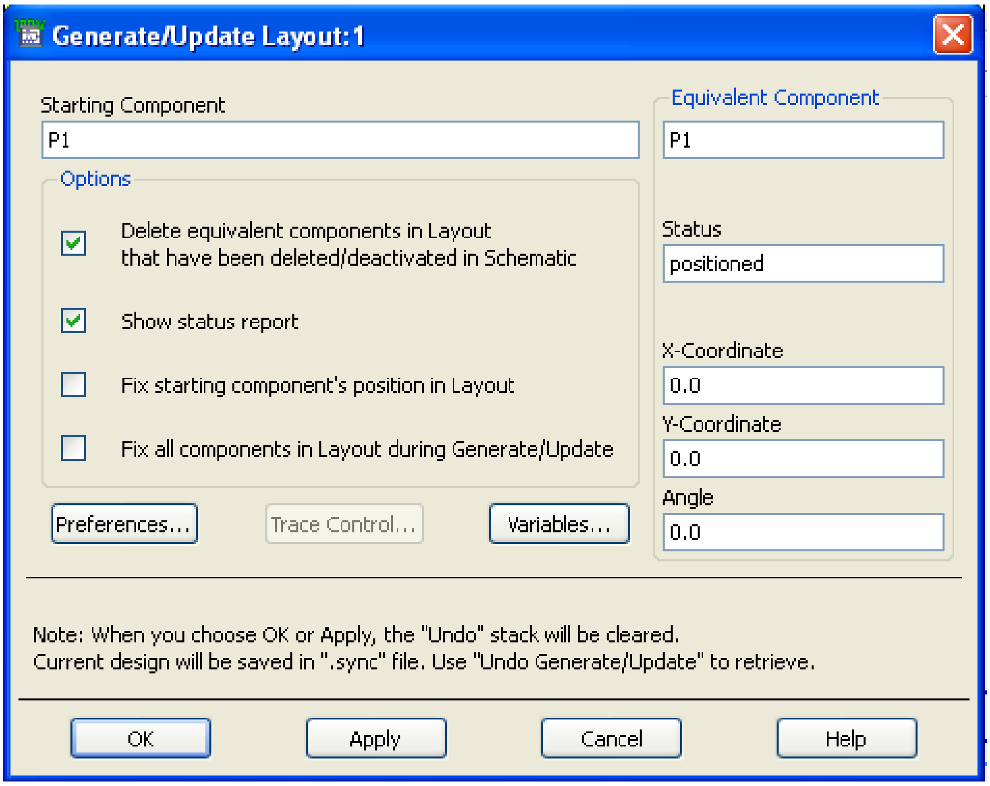


Figure 6.9 It provides to generate layout design.

After created the circuit, “Generate/Update Layout” is selected from the “Layout” menu in the top of schematic window. Then, another window is appeared as illustrated in Figure 6.9. It is “Generate/Update Layout” window. After making adjustments from this window, design is saved in ".sync" file. Another new window is appeared called as layout as shown in Figure 6.10.

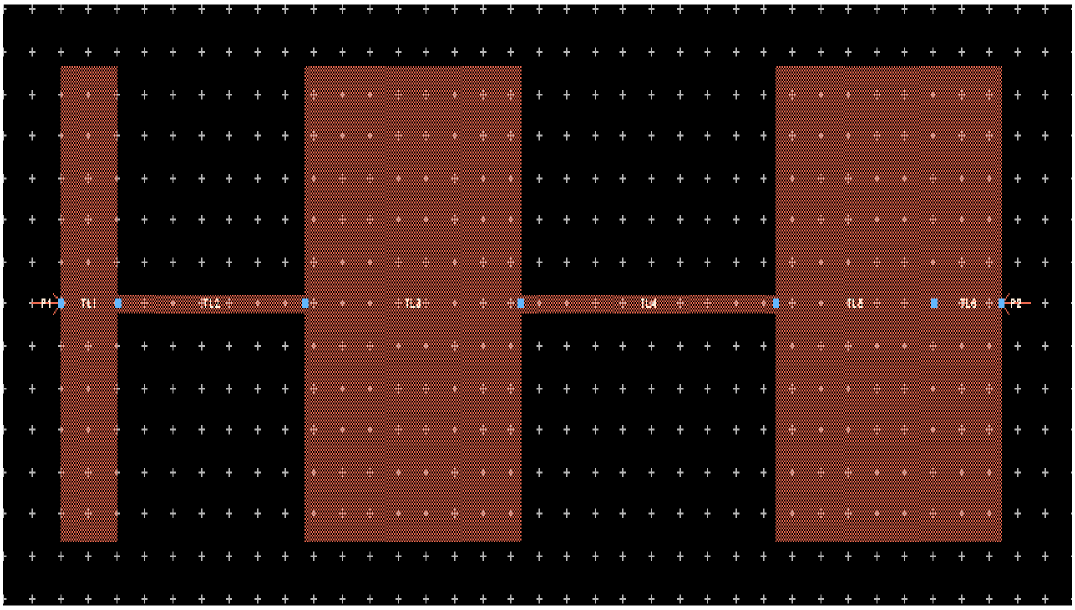


Figure 6.10 Layout window indicates equivalent components.

After generated the layout view, the next step to simulate the circuit is making chooses from "Momentum" menu. "Create/Modify" is selected under the part of "Substrate" in "Momentum" menu. Then a new window called as "Create/Modify Substrate" is opened. The name of substrate layers can be changed from this menu. It is also possible to adjust the thickness, permittivity and permeability. But this is not the only way to proceed. To take all this information from the current schematic design "Update from schematic" is chosen under the "Substrate" in "Momentum" menu. After setting these changes, it is seen that the values of thickness and other specifications such as permittivity and permeability are the same as set in MSub1 component. It is shown in Figure 6.11.

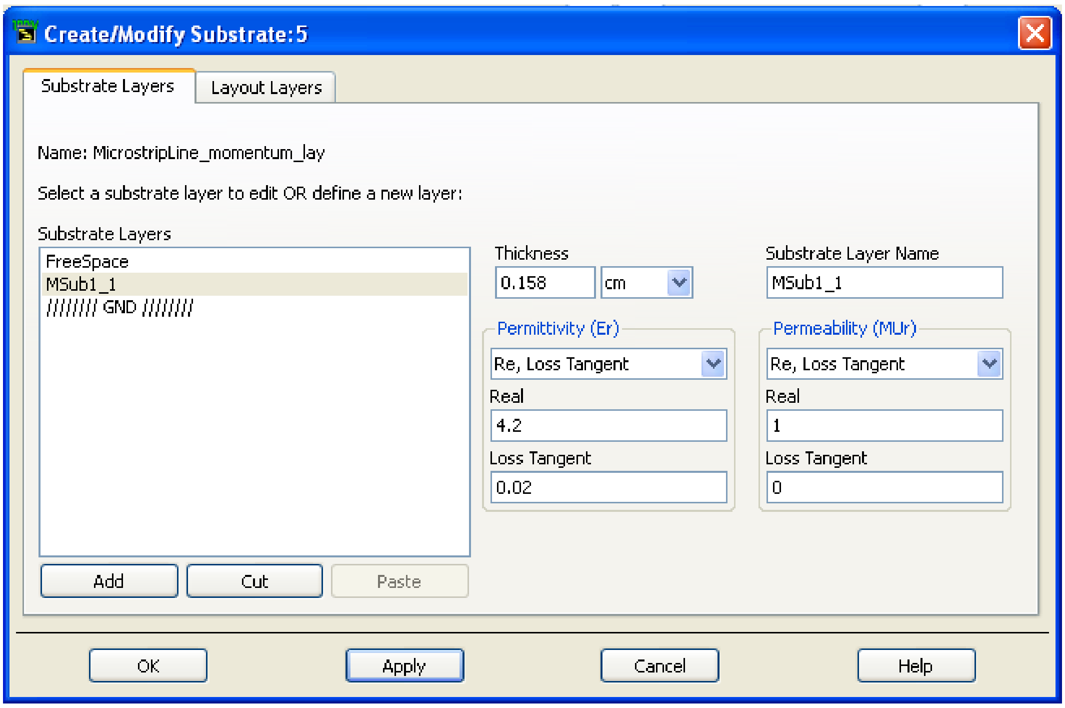


Figure 6.11 Adjustments of substrate layer are set from this window.

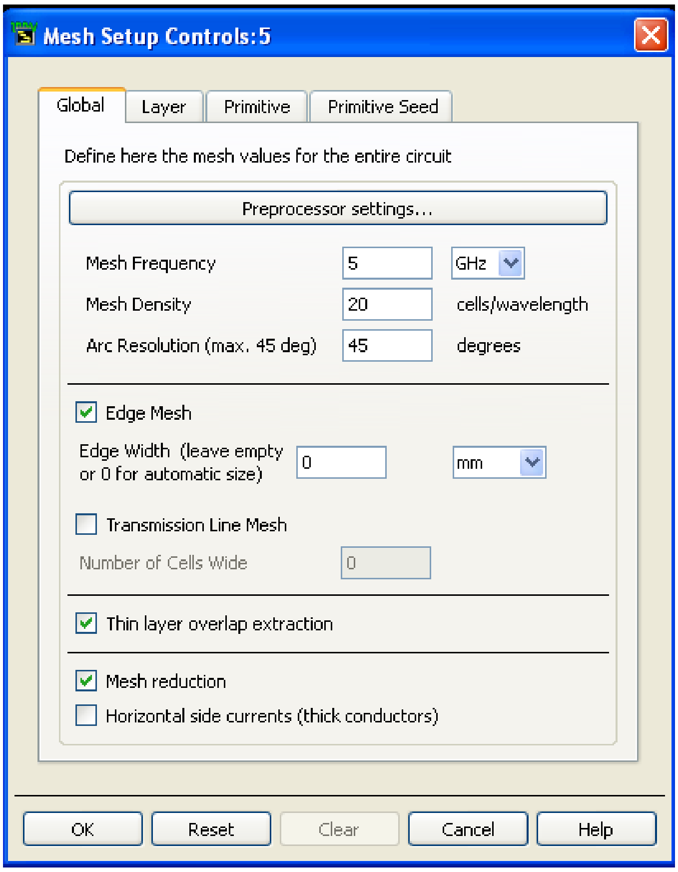


Figure 6.12The highest frequency in the circuit is chosen as mesh frequency.

The next setting is done from the part of “Setup” under the “Mesh” menu in “Momentum”. Mesh frequency is set from this menu. The highest frequency has to be entered for the mesh frequency. As it is seen in Figure 6.4, the highest frequency of this simulation is 5GHz. Setting of mesh frequency is done as shown in Figure 6.12. Start and stop frequency is set under the menu of “S-Parameter” in “Simulation” under the “Momentum”. After setting the frequency values, it should be updated such as shown in below window.

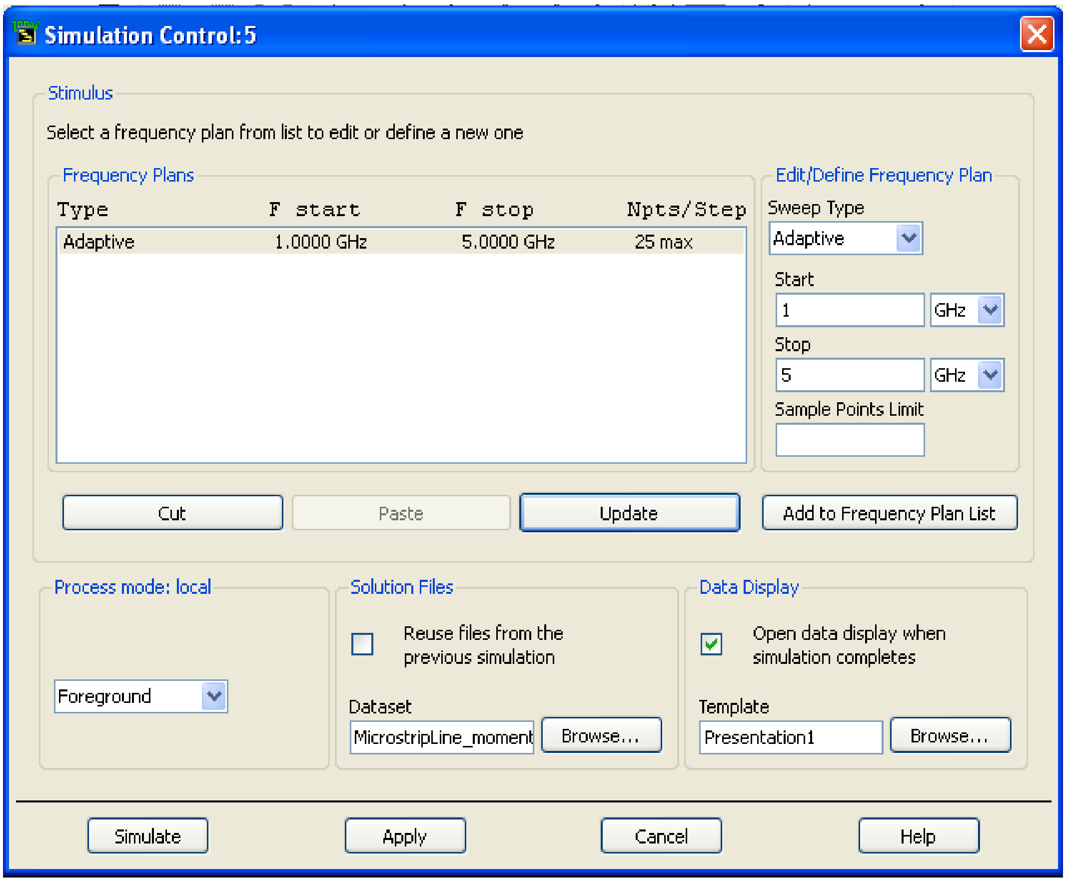


Figure 6.13 Frequency range is set in simulation control window before simulation.

Simulate button is pressed after updating the frequency range. Simulation is performed. Autoplot data display window is appeared as long as the simulation is completed. The full-wave simulator results as datasets for S(1,1), S(2,2), S(2,1) and S(1,2) are shown in data display window. All of these datasets are called as the response of low pass filter designed in Figure 6.10.

The plots of magnitude, phase and smith chart for S(2,1) are respectively shown in Figure 6.14 andFigure 6.15.

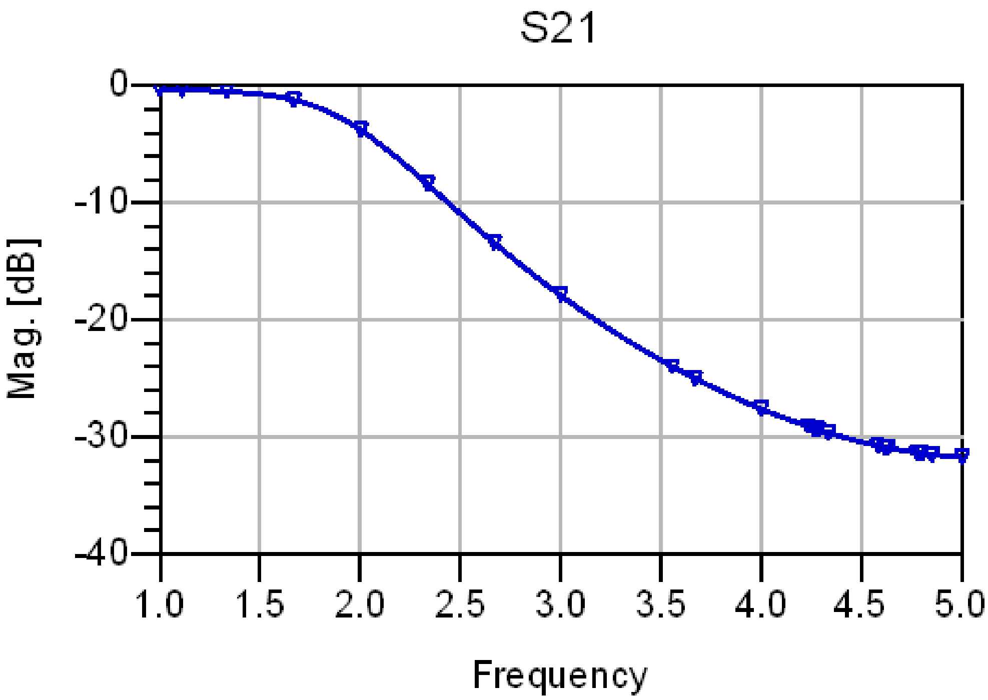


Figure 6.14 The magnitude (dB) versus frequency of S(2,1).

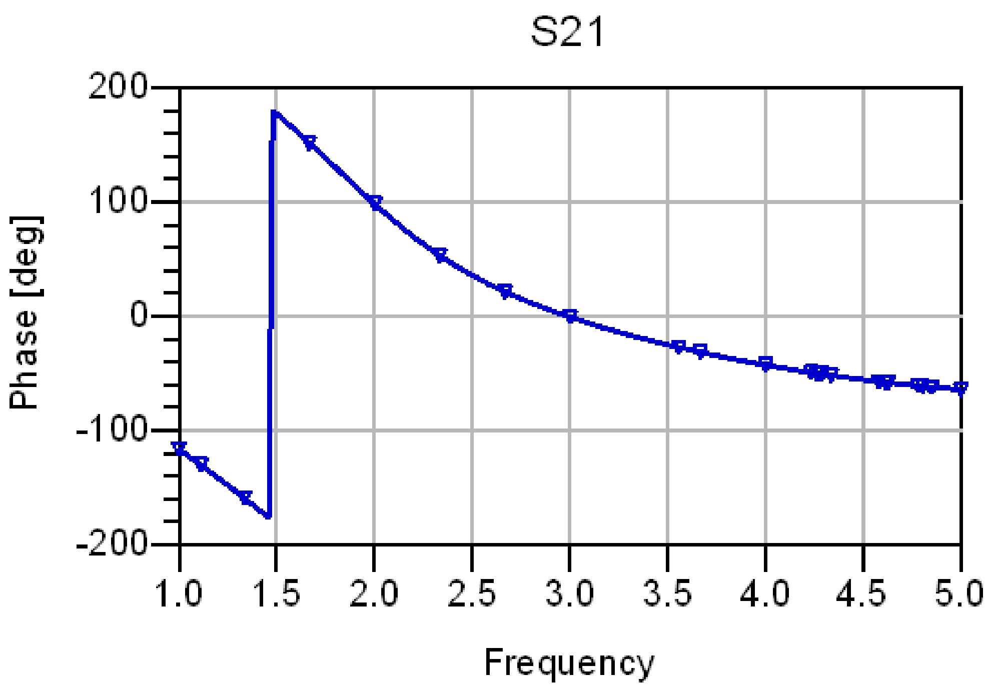


Figure 6.15 The phase (deg) versus frequency of S(2,1).

When designs get more complex, some differences between the schematic simulator and full-wave simulator emerges. As it is indicated at the beginning of this part, the results are more accurate from the results in schematic simulator. Because full-wave simulator referred as momentum simulator is based on the Method ofMomentsnumerical method.

6.3 Designing Class A Amplifier with Microstrip Lines in ADS Software

There are two ways to design and simulate Class A Amplifier in ADS software. First one is using schematic simulator; the second one is using the full wave simulator which is based on Methods-of-Moments.

6.3.1 Creating Class A Amplifier with Using Schematic Simulator

An example of Class A amplifier circuithas been performed with AC analysis in Figure 5.12 previous chapter. Similar circuit has been redesigned but with microstrip lines and S-Parameter analysis this time. Before adding microstrip lines the circuit has been simulated with S-Parameter. Hence AC source was removed from the circuit and port impedance terminations called “Term” have been chosen from “Simulation-S_Param” menu.

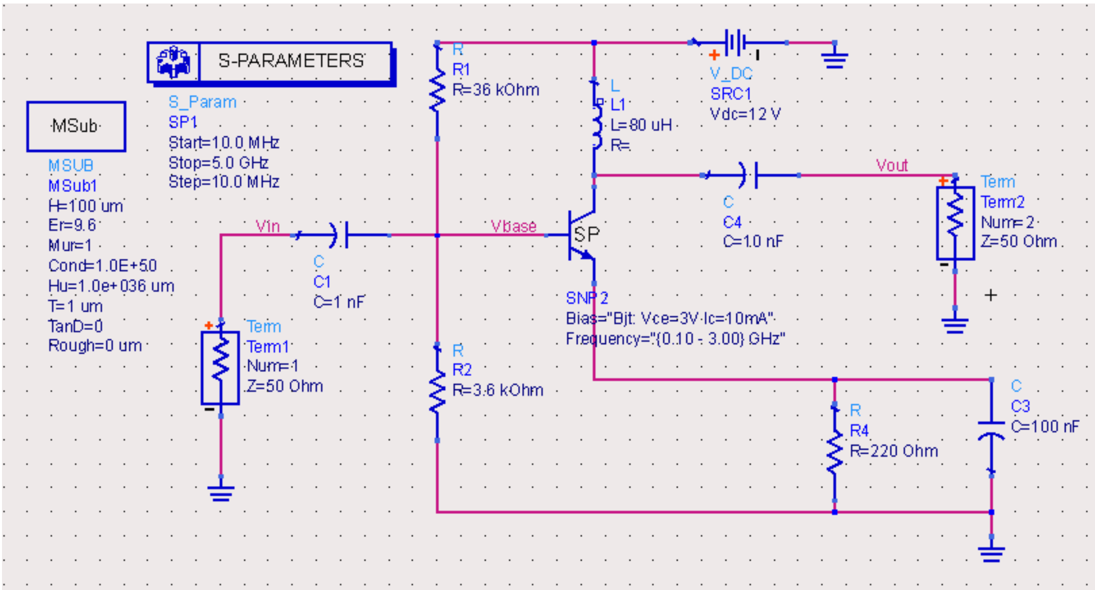


Figure 6.16 Class A amplifier circuit design with S-Parameter analysis.

Terminations with 50ohm value have been added to the input and output of the circuit shown in Figure 6.16. While simulating in schematic simulator, S-Parameter equations are used. After simulating the circuit, the gain response of the amplifier called as S(2,1)can be plotted in data display window. As it is seen from below figure, designed circuit has admissible high gain up to 2GHz approximately.

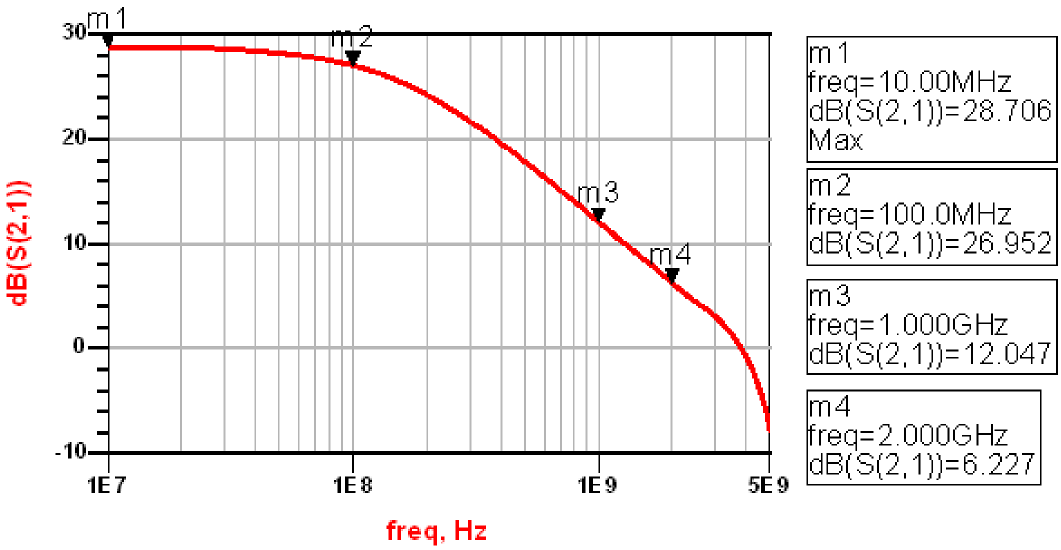


Figure 6.17 The magnitude (dB) versus frequency of S(2,1).

The next step is to perform the same Class A amplifier circuit with microstrip lines to display the electromagnetic effects. A realistic approach has been followed when adding microstrip lines to existing circuit. Cable connection between the voltage source and component is normally longer than other connections. The same case can be considered for the emitter and collector connections of transistor. Therefore three of microstrip lines have been added instead of these connections. A view of circuit schematic is shown in Figure 6.18. After adding MLIN components, MSUB is placed on the schematic. Substrate parameters are changed via MSUB component.

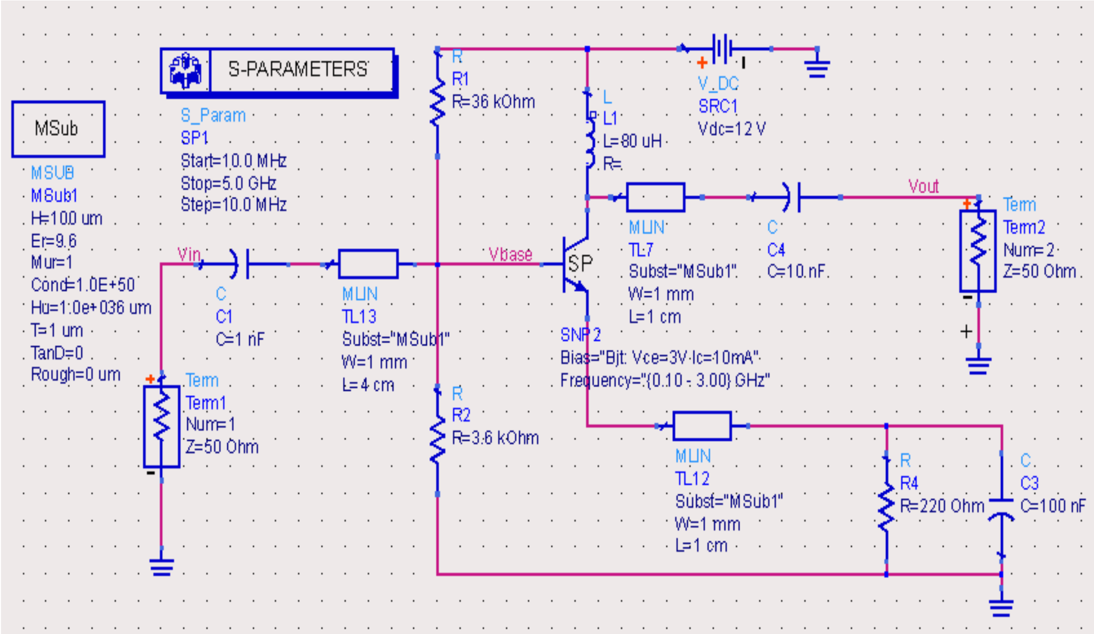


Figure 6.18 Class A amplifier circuit design with microstrip lines.

The response of circuit is shown in Figure 6.19 and 6.20. The amplifier design with microstrip line is more convenient to the design in practice. So acquired results are similar to real-like. As it is seen from the Figure 6.19,the gain decreases and the bandwidth gets smaller as compare with the response of the circuit shown in Figure 6.16. The circuit designed with microstrip lines has admissible high gain up to 500MHz approximately as it was 2GHz in Figure 6.17.

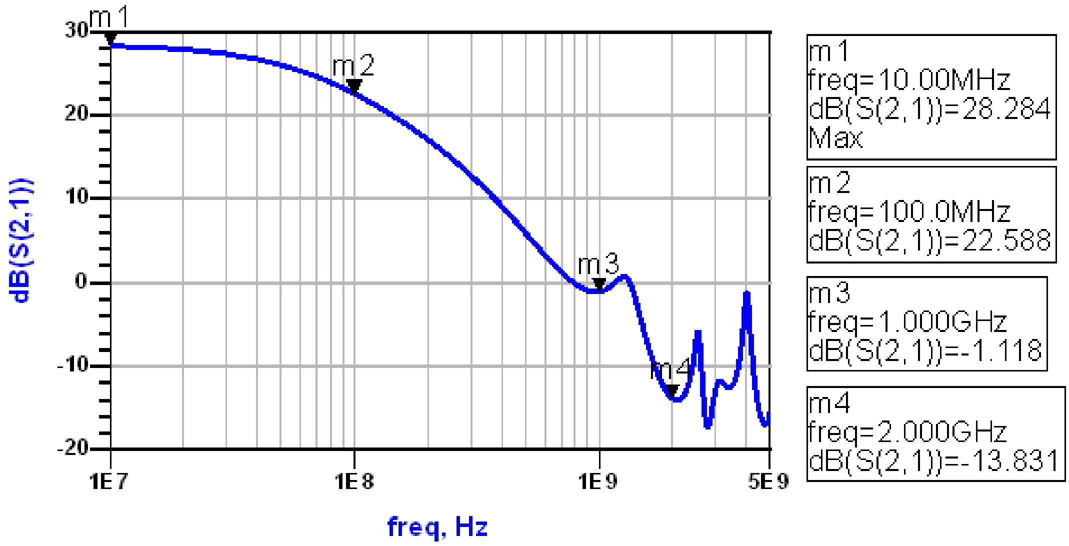


Figure 6.19 The gain response of the Class A amplifier designed with microstrip lines.

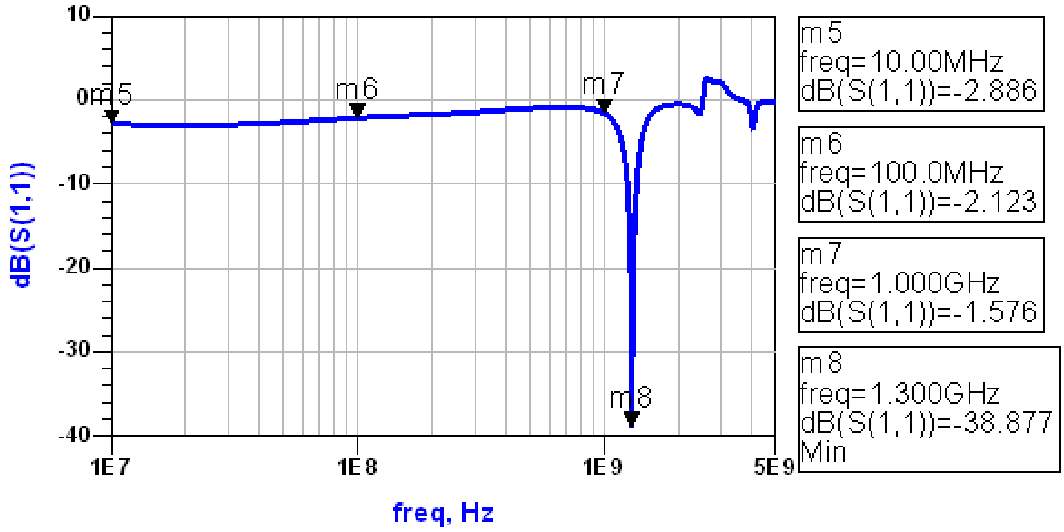


Figure 6.20The magnitude (dB) versus frequency of S(1,1).

As a conclusion, it will be useful to see the response of the circuits together. The only difference between two circuit designs is existence of microstrip lines. Therefore the effect of the microstrip line could be seen significantly.It is appreciable from the designs that electromagnetic effects have been occurred at high frequencies especially higher than 1GHz. The disturbance signal might be originated from parasitic effects and non ideal behavior of circuit components such as transmission lines.The responses of the circuitsdesigned in Figure 6.16 and Figure 6.18 are shown juxtaposed in Figure 6.21.

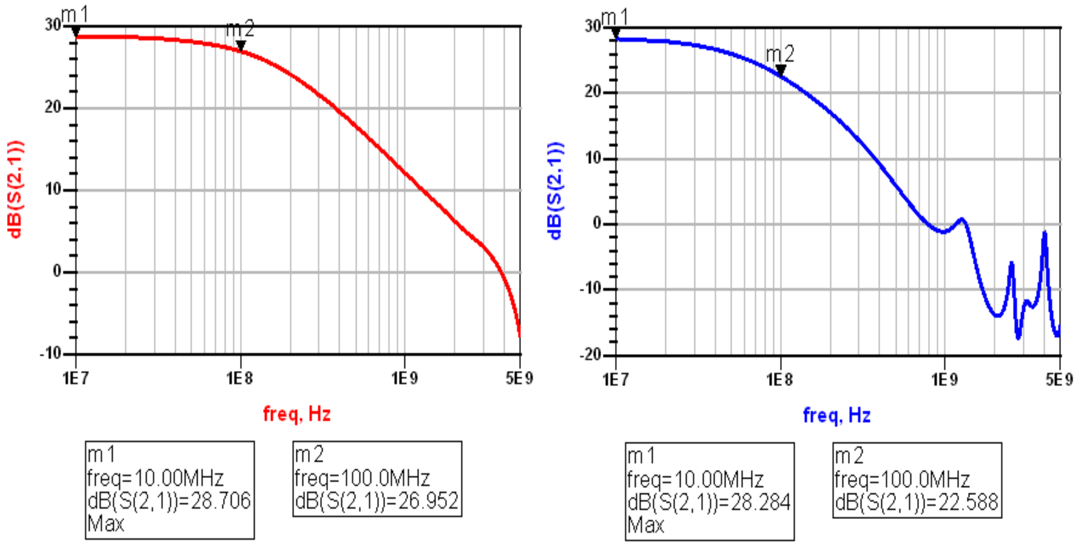


Figure 6.21Thegain responses of Class A amplifier circuit designs.

6.3.2 Creating Class A Amplifier with Using Full-Wave Simulator

The second step is performing the Class A amplifier circuit with the full-wave simulator. The circuit design in schematic window should be saved as in another name. Ports are inserted instead of the source and load resistor. Port icon is achieved from the top of the menu in schematic window. The new created circuit can be seen in Figure 6.22.

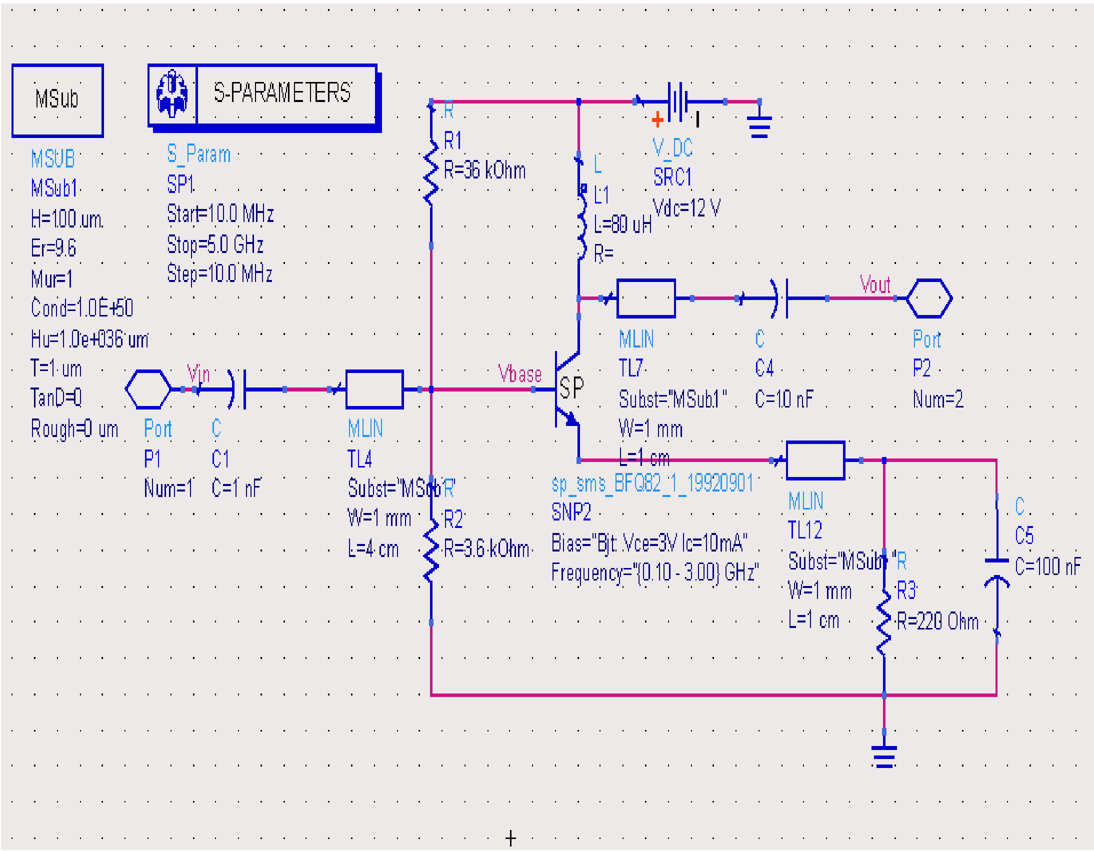


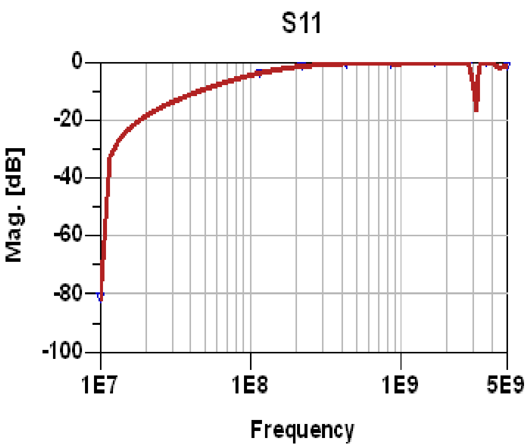
Figure 6.22Ports are inserted to the amplifier circuit to simulate with full-wave simulator.

After created the circuit, “Generate/Update Layout” is selected from the “Layout” menu in the top of schematic window. “Generate/Update Layout” window is appeared. After making adjustments from this window, design is saved in “.sync” file. Another new window is appeared called as layout. “Create/Modify” is selected under the part of “Substrate” in “Momentum” menu.

The name of substrate layers can be changed from “Create/Modify Substrate” menu. It is also possible to adjust the thickness, permittivity and permeability. To take all this information from the current schematic design “Update from schematic” is chosen under the “Substrate” in “Momentum” menu. After setting these changes, it is seen that the values of thickness and other specifications such as permittivity and permeability are the same as set in MSub1 component.

The next setting is done from the part of “Setup” under the “Mesh” menu in “Momentum”. Mesh frequency is set from this menu. The highest frequency has to be entered for the mesh frequency. Start and stop frequency is set under the menu of “S-Parameter” in “Simulation” under the “Momentum”. After setting the frequency values, it should be updated such as shown in Figure 6.13 again.

The full-wave simulator results as datasets for S(1,1) and S(2,1) which are called as the response of Class A amplifier designed in Figure 6.22are shown in data display window. The shapes of plots are similar as they are shown in below figures. The electromagnetic effects are considerable at high frequencies especially higher than 1GHz. The parasitic effects and unexpected behavior of circuit components such as microstrip lines which are located at three parts in this circuit are important parameters. Since, the response of circuit changes especially at high frequencies for instance the gain decreases. The plots of magnitude and phase for S(1,1) and S(2,1) are respectively shown in below figures.



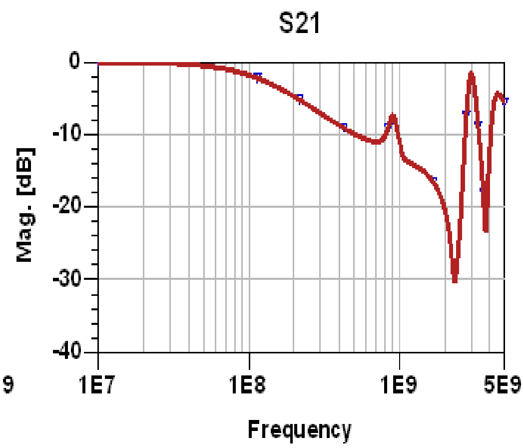


Figure 6.23 The magnitude (dB) versus frequency of S(1,1) and S(2,1) datasets.

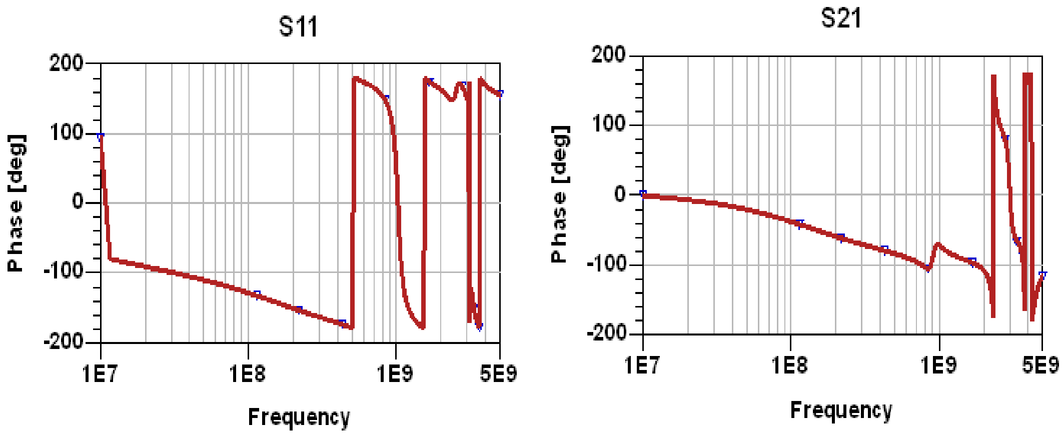


Figure 6.24 The phase (deg) versus frequency of S(1,1) and S(2,1) datasets.

6.4 Short Circuit and Open Circuit Effects of Microstrip Lines

In low frequencies short circuited microstrip line shows inductance effect until the resonance frequency. A circuit has been created via schematic window of ADS software to simulate the inductance effect of short circuited microstrip lines as shown in Figure 6.22.In this example I_Probe component has been chosen from the “Probe Components” menu. It has been used to measure the current flowing from the short circuited line.

Magnetic field occurs around the conducting wire that the current flows inside. If this current is an ac current, a voltage which is reacting to the changes of current occurs. That is called as inductance effect.

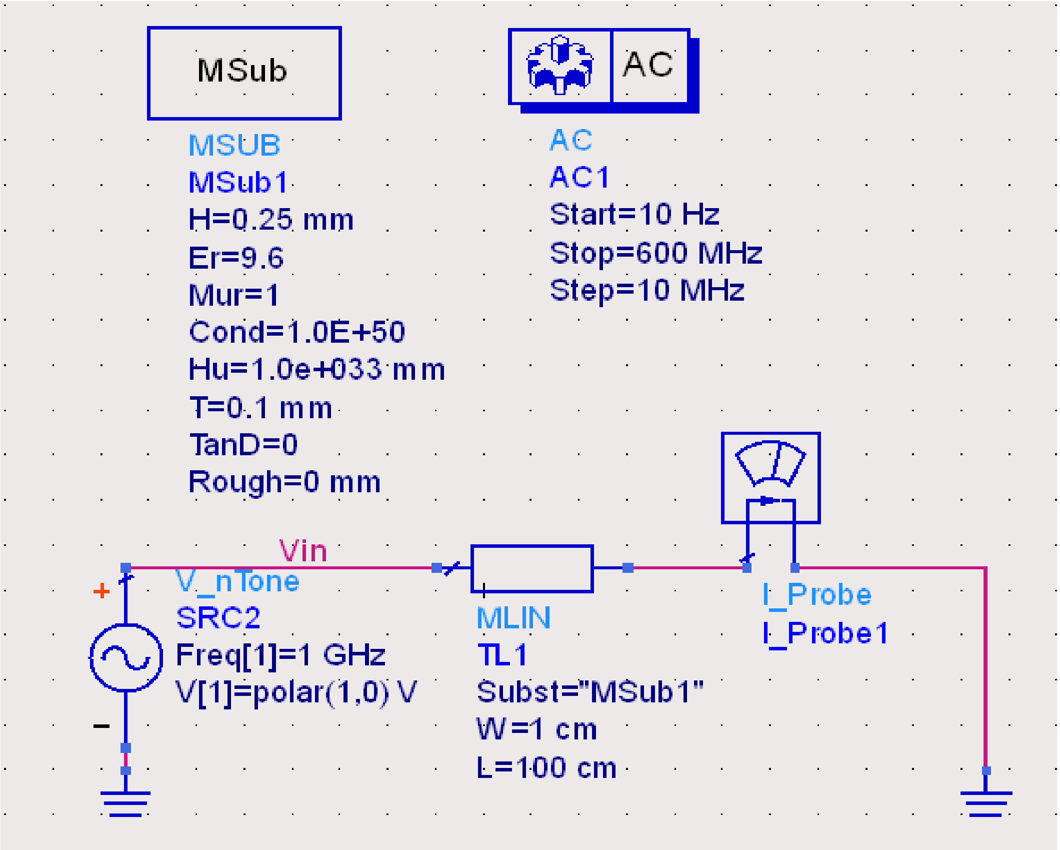


Figure 6.22 Short circuited design of microstrip line.

Magnitude, phase and dB graphs of current probe versus frequency are shown in the following figures respectively.

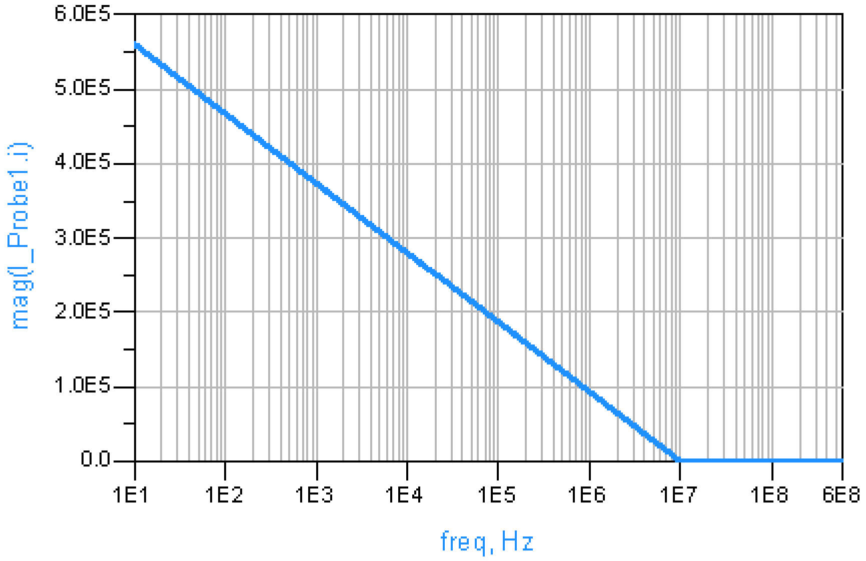


Figure 6.23Magnitude of I_Probe versus frequency.

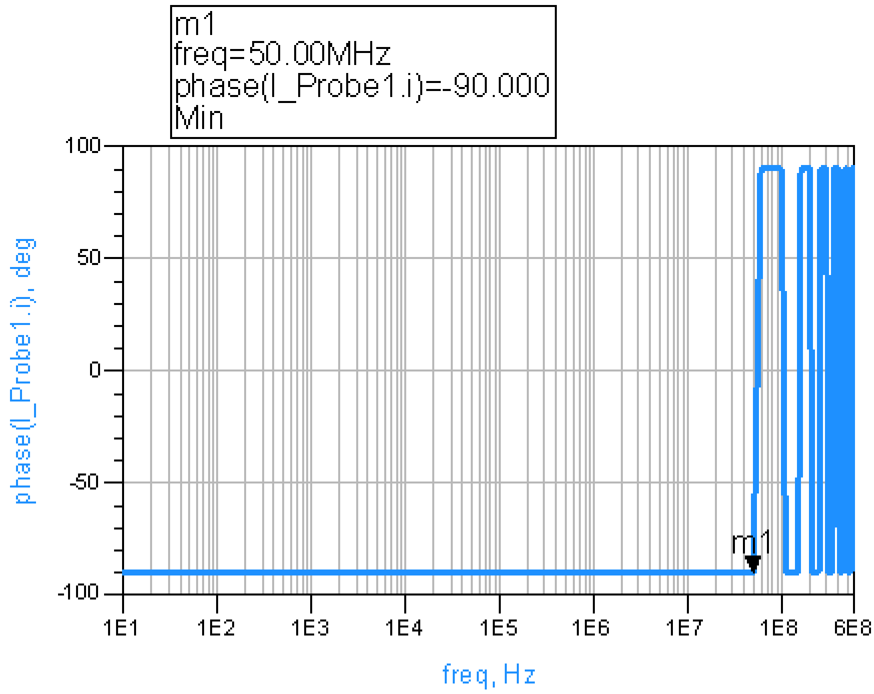


Figure 6.24 Phase of I_Probe versus frequency.

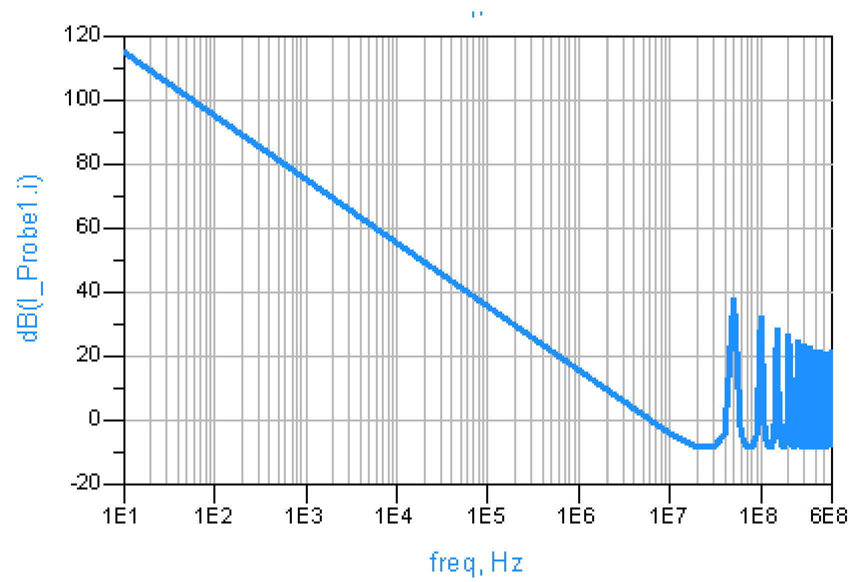


Figure 6.25 Decibel of I_Probe versus frequency.

The graphs in Figure 6.23, 6.24 and 6.25 are all the response of the circuit that shows inductive effect and behaves as an inductor until the determined frequency. That is the resonance frequency. It is almost 50MHz for this example. After the resonance frequency, the behavior of the microstrip line changes. The circuit sometimes shows inductor effect and sometimes capacitor effect.

In low frequencies open circuited microstrip line shows capacitance effect until the resonance frequency. A circuit has been created via schematic window of ADS software to simulate the capacitance effect of open circuited microstrip lines as shown in Figure 6.26. In this example I_Probe component has been chosen from the "Probe Components" menu. It has been used to measure the current flowing between two microstrip lines which are open circuit.

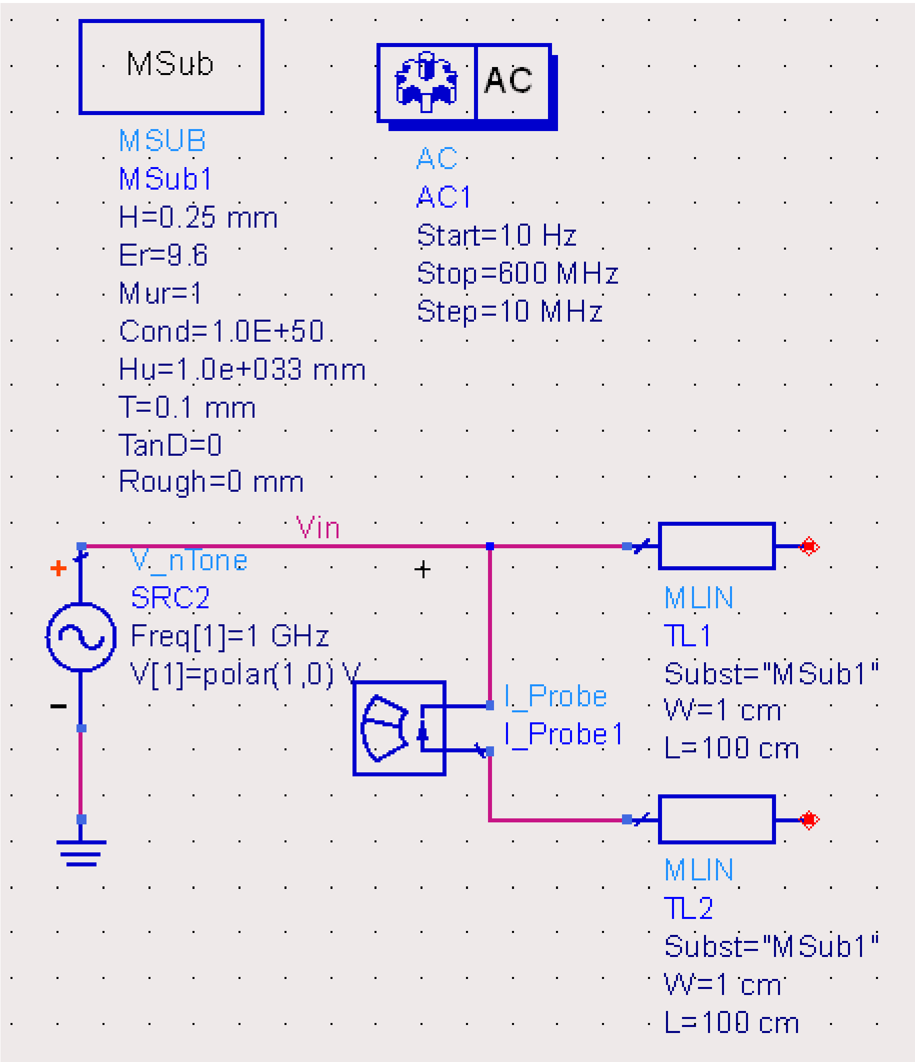


Figure 6.26 Open circuited design of microstrip line.

Electric field occurs around the conducting wires which are parallel to each other. This effect is related with the width and dielectric material of conductors. The distance between the conductors is also an important parameter for the capacitance effect.

Magnitude, phase and dB graphs of current probe versus frequency are shown in the following figures respectively.

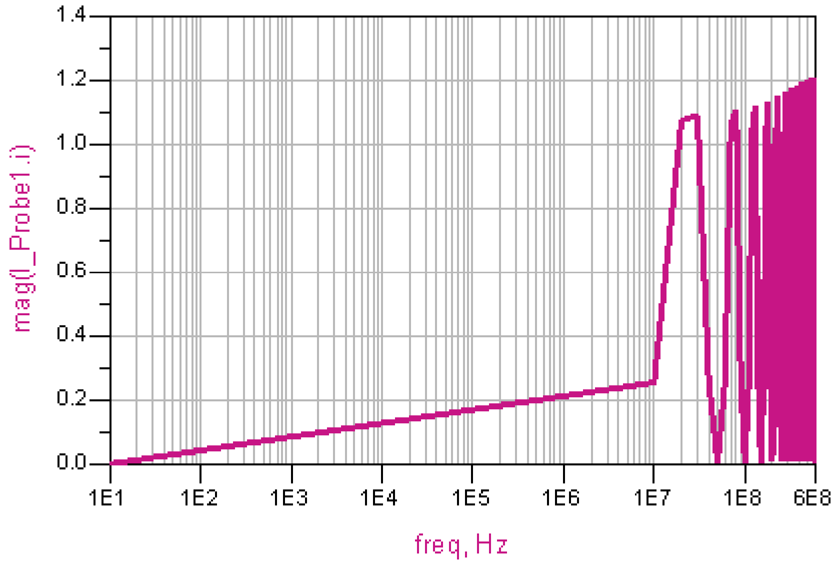


Figure 6.27 Magnitude of I_Probe versus frequency.

The plots in Figure 6.27, 6.28 and 6.29 are all the response of the circuit that shows capacitance effect and behaves as a capacitor until the determined frequency. That is the resonance frequency. After the resonance frequency, the behavior of the microstrip line changes. The circuit sometimes shows inductor effect and sometimes capacitor effect.

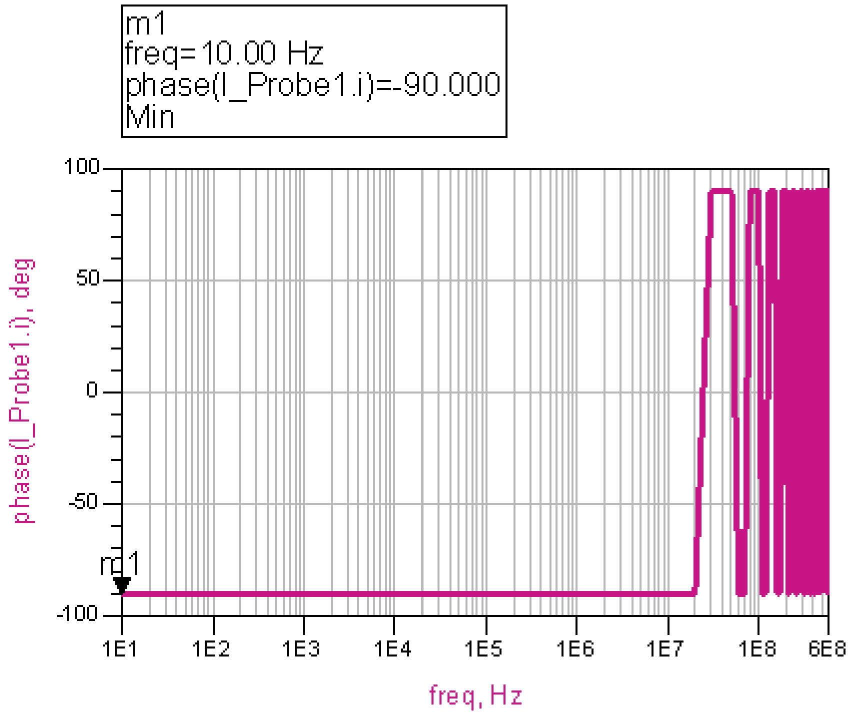


Figure 6.28 Phase of I_Probe versus frequency.

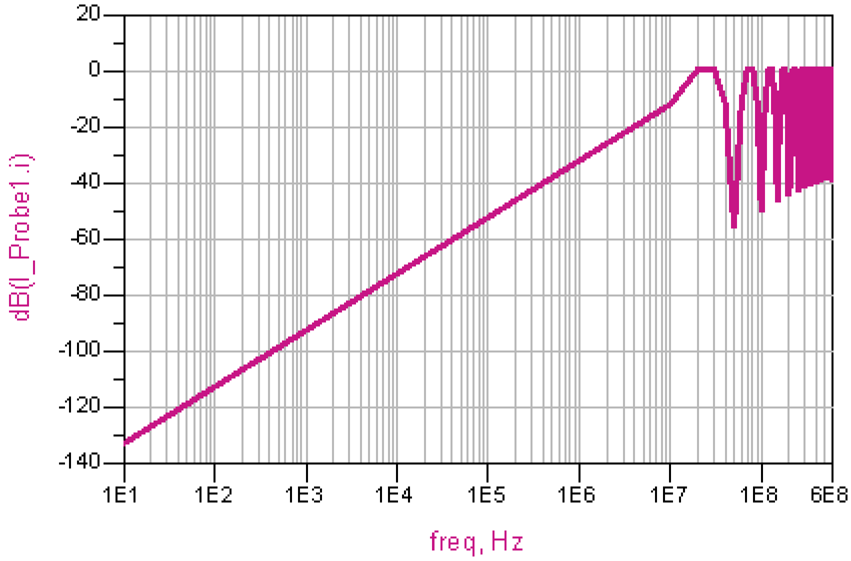


Figure 6.29 Decibel of I_Probe versus frequency.

CHAPTER SEVEN
CONCLUSIONS

Circuit simulators are indispensable tools for circuit analysis and design. On the other hand well known spice software packages only take into account circuit component models. However, especially at high frequencies layout of the circuit and wiring techniques used for realizing the circuit have important effects on the circuit performance. For example although a spice simulator may give a result that indicates the circuit will work properly. Actually the same circuit may fail in practice. Due to this issue some elaborate software packages are available today and this software packages take into account circuit layout and physical structure of the circuit. Agilent ADS is one of the software tools in this category.

In this study a radio frequency amplifier (single transistor class-A)was designed and simulated using ADS software. The input frequency was selected between 10MHz and 5GHz.High gain with wide bandwidth was obtained from simulation results. Low noise amplifier circuit examples were designed and simulated using ADS software. While using HBFP-0450 transistor the operating frequency was selected between 1GHz and 3GHz. Maximum gain was approximately obtained at 1.5GHz. The same circuit was redesigned with Murata components to see the more accurate results. Murata components which have modifiable parameters let circuit operate more suitable in comparison with practical appliances. Parasitic effects of components may also be added and changed with this component library. It is observed that the gain of the low noise amplifier and the frequency at which the maximum gain decreases when the circuit is designed with Murata components.

In later parts of this study a low pass filter operating in the range of 1GHz to 5GHz, was designed with microstrip lines to observe the effects of transmission line causing electromagnetic disturbances. The low pass filter was designed and simulated by two ways of usage microstrip lines in ADS software. One of them is using schematic simulator; the other is using the full-wave simulator which is based on Methods of Moments. While simulating in schematic simulator, S-Parameter equations are used. The results of the schematic simulator and full wave simulator are nearly the same, but when designs get more complex it is possible to see some differences. The expected and acceptable fact is that the results of full wave simulator are more accurate from the results in schematic simulator. Because full-wave simulator referred as momentum simulator is based on the Method of Moments numerical method.

The radio frequency amplifier (single transistor class-A) was reformed with microstrip lines and S-Parameter analysis this time. In this example microstrip lines have been added in place of the connections which may be normally longer than others. The obtained gains at 100MHz, 1GHz and 2GHz are compared with the gain obtained from the circuit which is designed without microstrip lines. The amplifier design with microstrip line is more convenient to the design in practice. So acquired results are similar to real-like. It is observed that the gain and bandwidth decrease due to layout parasitics. The electromagnetic effects are considerable at high frequencies especially higher than 1GHz. It is appreciable that the parasitic effects and unexpected disturbance behavior of transmission lines are important parameters for high frequency circuit design. The same circuit has been designed with full-wave simulator and similar results have been obtained.

Short circuited and open circuited designs have been used to consider the high frequency effect on microstrip lines. In low frequencies short circuited microstrip line shows inductance effect and behaves as an inductor until the resonance frequency. Similarly, in low frequencies open circuited microstrip line shows capacitance effect and behaves as a capacitor until the resonance frequency. After the resonance frequency, the behavior of the microstrip line changes. The circuit sometimes shows inductor effect and sometimes capacitor effect.

As a result of this study it is possible to see that circuit simulators help to improve efficiency of circuit design. After designing and simulating the circuit in software program, it is ensured that the circuit will operate satisfactorily or electromagnetic effects should be reduced by re-designing physical layout of the circuit.

Accurate placement of circuit elements, selecting the suitable operating frequency and having knowledge about the requirements of electromagnetic compatibility design help to minimizing the product cost and maintaining the function quality of systems.

In this study electromagnetic effects of high frequency circuit design have been considered with the help of microstrip lines. Thus the importance of cable and connectors is pointed out in circuit layout.For future work, reduction techniques of electromagnetic interference might also be studied on with designing circuit in ADS software. For instance, applying filters to the circuit or increase the shielding efficiency are also acceptable to reduce the electromagnetic disturbance effects. These solution ways might be simulated in ADS software and implemented as an advance stage.

It is also possible to realize radio frequency circuit simulation via EMDS (Electromagnetic Design System) tool integrated into ADS software. This tool enables to make three dimensional electromagnetic simulations via using Finite Element Method. It also allows visualization of electric and magnetic fields in simulation. Radio frequency circuit design and simulation shall also be studied and considered with using EMDS tool to get more accurate results and scrutinizingelectromagnetic effects as a progression to this thesis.

REFERENCES

Agilent Technologies, (n.d.). *Agilent AN 1287-1 Understanding the FundamentalPrinciplesof Vector Network Analysis*, Retrieved February 1, 2012, fromhttp://cp.literature.agilent.com/litweb/pdf/5965-7707E.pdf

Agilent Technologies, (2012). *Agilent EEsof EDA Advanced Design System*.Retrieved January 20, 2012, from http://cp.literature.agilent.com/litweb/pdf/5988-3326EN.pdf

Agilent Technologies,(n.d.).*Advanced Design System (ADS).* Retrieved January 20, 2012,from http://www.home.agilent.com/agilent/product.jspx?cc=US&lc=eng&ckey=1297113&nid=-34346.0.00&id=1297113

Agilent Technologies,(n.d.). *Simulation and Optimization Controllers*, Retrieved January 28, 2012, from http://cp.literature.agilent.com/litweb/pdf/ads2008/adstour/ads2008/Simulation_and_Optimization_Controllers.html

Agilent Technologies, (2008).*Test & Measurement Application Note 95-1 S-Parameter Techniques,* Retrieved November 14, 2011, fromhttp://cp.literature.agilent.com/litweb/pdf/5989-9273EN.pdf

Arı, N. &Özen, Ş.(2008). *ElektromanyetikUyumluluk*(1sted.). Ankara: Palme Yayıncılık

Bowick, C., Blyler, J., Ajluni, C. (2007). *RF Circuit Design* (2nd ed.).United States of America:Elsevier

Celozzi, S., Araneo, R., Lovat, G. (2008).*Electromagnetic Shielding*(2nd ed.).United States of America: John Wiley & Sons, Inc.

Computer Simulation Technology AG, (2010).*CST Microwave Studio*. Retrieved May 5, 2012, from http://eee.guc.edu.eg/Courses/Communications/COMM905%20Advanced%20Communication%20Lab/MWS_Tutorials.pdf

Electropedia, (1992). Retrieved September 10, 2011, fromhttp://www.electropedia.org/iev/iev.nsf/display?openform&ievref=702-08-66

EM Talk, (n.d.).*ADS Tutorial 2 S-Parameter Simulation: Momentum*. (2011). Retrieved May 5, 2012, from http://www.emtalk.com/ads_tut_2.htm

Gonschorek, K. & Vick, R. (2009), *Electromagnetic Compatibility for Device Design and System Integration* (1st ed.). Germany: Springer-Verlag Berlin Heidelberg

Hubing, T.H. (n.d.). *Non-Ideal Behavior of Components*, Retrieved August 11, 2012, fromhttp://www.cvel.clemson.edu/Presentation_Slides/Non-Ideal_Behavior_of_Components.pdf

Hubing, T., &Orlandi, A. (n.d.),*A Brief History of EMC Education*Retrieved February 10, 2012, from http://www.clemson.edu/ces/cvel/pdf/zur05-095.pdf

Joffe, E.B. & Lock, K. (2010), *Grounds for Grounding* (1st ed.). USA: John Wiley & Sons, Inc. Publication.

Maloratsky,Leo G.(2000). *Reviewing The Basics Of Microstrip Lines*.Retrieved May10, 2012, fromhttp://paginas.fe.up.pt/~hmiranda/etele/microstrip_basics.pdf

Morrison, R. (2002). *The Fields of Electronics: Understanding Electronics Using Basic Physics.* (1sted.). U.S.A: John Wiley & Sons, Inc. Publication.

Paul, C.R. (2006). *Introduction to Electromagnetic Compatibility* (2nd ed.).U.S.A: John Wiley & Sons, Inc. Publication.

Sevgi, L. (2006). *TasarımdanÜretimeElektromanyetikUyumluluk* (1sted.). İstanbul: EksenYayıncılık

Sischka, F.(2002), *Basics of S-Parameters.*Retrieved February 5, 2012, fromhttp://tesla.unh.edu/courses/ece711/refrense_material/s_parameters/1SparBasics_1.pdf

Williams, T., &Armstrong, K. (2000).*EMC for Systems and Installations* (1sted.). Great Britain: BiddiesLtd, GuildfordandKing'slynn

Yarman, B.S.&Şengül, M.(2008).Chapter 12 Analysis and Optimization ofMatching Network-I.*Design of Ultra Wideband Antenna Matching Networks*(1st ed.) (281-292). Springer Science+Business Media B.V.

Yarman, B.S. (2010). *Design of Ultra Wideband Power Transfer Networks* (1st ed.). United Kingdom: John Wiley & Sons Ltd.