A PROJECT REPORT

ON

"DESIGN & DEVELOPMENT OF LOW NOISE AMPLIFIER (LNA) FOR GMRT RADIO TELESCOPE"

BY

Mr. SUJIT B. DHARMPATRE

M. Tech. (Electronics & Telecomm.- Microwave)

(MO423M16)

Govt. College of Engineering, Pune

PUNE - 411005.

Year : 2005-2006

Semester IV

---------------------------------- Sponsored By --------------------------------

GMRT OBSERVATORY, NCRA,

TATA INSTITUTE OF FUNDAMENTAL RESEARCH, PUNE

CERTIFICATE

This is to certify that

Mr. SUJIT B. DHARMPATRE

(MO423M16)

Fourth semester M. Tech. (Electronics & Telecomm.- Microwave) Engineering has duly submitted a Project Report on

"DESIGN & DEVELOPMENT OF LOW NOISE AMPLIFIER (LNA) FOR GMRT RADIO TELESCOPE"

During the academic year 2005 – 2006 for partial fulfillment of the Requirment for the award of Master of Technology course of College of Engineering, Pune done at Giant Meter-wave Radio Telescope (TIFR)

PROF. V. K. KOKATE
Internal Guide,
E & Tc Engg. Dept.
Director,COEP Pune

PROF. DR. (MRS.) M. A. JOSHI
Head Of the Department,
E & Tc Engg. Dept.
COEP, Pune

MR. A. PRAVEEN KUMAR
External Guide,
Group Co-ordinator,
RF & FIBER – OPTIC GROUP, GMRT, NARAYANGAON.
TATA INSTITUTE OF FUNDAMENTAL RESEARCH, PUNE.

<u>EXAMINATION APPROVAL SHEET</u>

The project report entitled,

“DESIGN & DEVELOPMENT OF LOW NOISE AMPLIFIER (LNA) FOR GMRT RADIO TELESCOPE”

By

Sujit B. Dharmpatre

(MO423M16)

is approved for the degree of **MASTER OF TECHNOLOGY(ELECTRONICS AND TELECOMMUNICATION ENGINEERING – MICROWAVE)** of Govt. College of Engineering, Pune- 411005.

Examiners

1. Chairman :
2. External Examiner :
3. Internal Examiner :
4. Supervisor :

Date:

Place:

ACKNOWLEDGEMENT

It gives me great pleasure to express my deep sense of gratitude and indebt ness to my guide – Prof. V. K. Kokate for his valuable guidance, constant encouragement and kind cooperation in all the phases of this seminar work & completion of this First Dissertation Report -

“Design & Development Of Low Noise Amplifier (LNA) For GMRT Radio Telescope”.

I am also thankful to Dr. Mrs. M. A. Joshi , HOD, E & Tc Engineering Dept for allowing me to work on this project. I specially thankful to Prof. A. M. Sapakal who gave me valuable guidance & encouragement in my project work.

I am extremely grateful to A. Praveen kumar (Group Co-ordinator,RF & Fiber-Optic Dept., GMRT, Khodad, Narayangaon) for giving me useful guidance, encouragement and valuable suggestions in the work and providing me all the facilities at GMRT, Khodad, Narayangaon .

My thanks are also to Mr. Anil Raut & Mr. Bhalerao & all Staff of GMRT Khodad, Narayangaon for their excellent cooperation and to all those who helped me directly or indirectly in completing this project.

Mr. Sujit B. Dharmpatre

CHAPTER 1

THE GIANT METER-WAVE RADIO TELESCOPE AND THE SIGNALS IT RECEIVES

GMRT(Giant meterwave Radio Telescope) is a Radio Telescope consisting of 30 parabolic sturable dish antennas, each 45 m in diameter, installed across the region of about 25 km in diameter, near the village Khodad, 90 km north of pune. It has been designed to operate at a range of frequencies form 50 MHz to 1450 MHz. GMRT is among the most powerful Radio Telescopes in the world at meter wavelength, established for frontline research in Astronomy and Astrophysics. Radio astronomy is the study of sky at radio wavelength. The field of radio astronomy was started in 1923, when Karl Jansky discovered that his antenna was receiving radiation from outside the Earth's atmosphere. He noticed that this radiation appear at the same sidereal time on different days and that its source must hence lie far outside the solar system. Further observations enabled him to identify this radio source as the center of the galaxy.

The dominant sources seen in the radio sky are the Sun, Supernova remnants, radio galaxies, pulsars etc.

Radio signal means the record of the electric field received at a point on the earth from a source of radio waves. This signal is naturally random in nature. A fundamental property of the radio waves emitted by cosmic sources is that they are stochastic in nature. i. e. electric field at earth due to distant cosmic source can be treated as a random process.

The whole goal of the radio astronomy is to receive, process, and interpret these cosmic signals.

By international agreement, the radio spectrum is allocated to different users. Radio astronomy has a limited number of protected bands where no one else is permitted to radiate and most radio telescopes work only at these protected frequencies.

Radio waves from the distant cosmic source impinge on the antenna and create a fluctuating voltage at the antenna terminals. This voltage varies at the same frequency as the cosmic electro-magnetic wave, referred to as Radio Frequency (RF). The voltage is first amplified by the front end (or RF) amplifier. The signal is weakest here, and hence it is very important that the amplifier introduce as little noise as possible. Front end amplifiers hence usually use low noise solid state devices like LNA (or High Electron mobility transistors – HEMTs, which satisfies the requirements of the LNA) often cooled to liquid helium temperature.

1.1 The Giant Meter-Wave Radio Telescope :

Radio telescope will have to receive extremely low level RF signals emitted by the celestial radio sources. Therefore radio astronomical receiver must have very low noise figure. Receiver front ends are built to have low noise figure & high gain. The low noise amplifier is incorporated as the first stage of the receiver front end.

A radio telescope in the simplest form consists of three components –

(1) An antenna which receives radiation from a small region of sky.

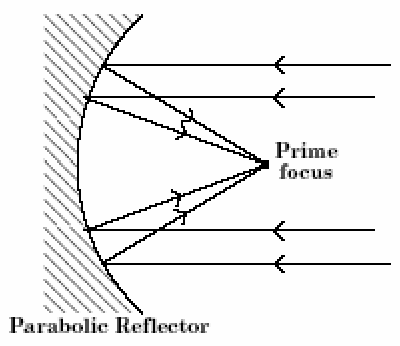


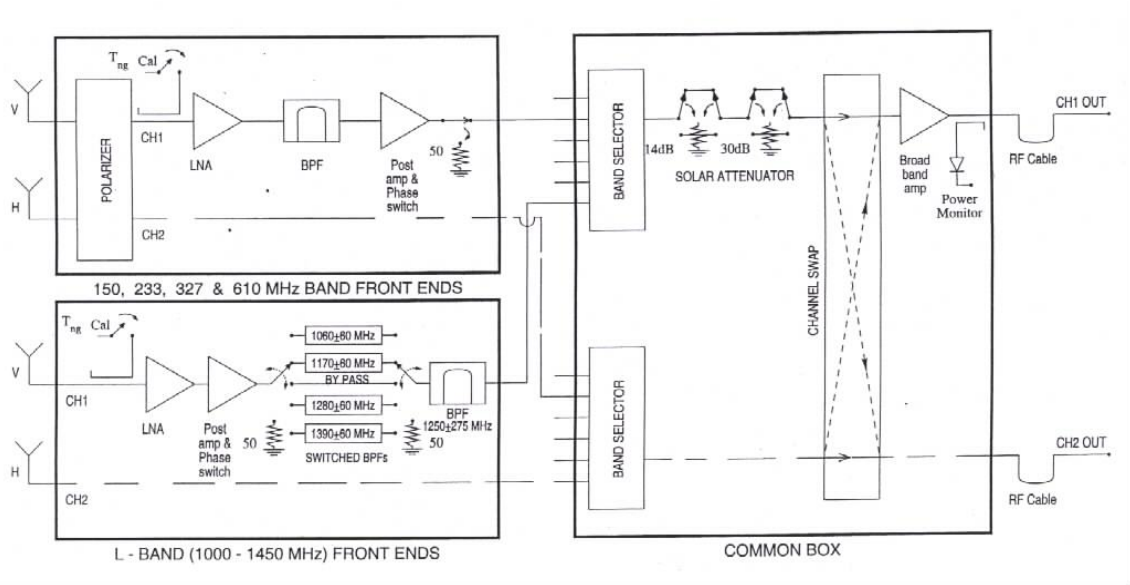
Fig. 1.1- Parabolic reflector

(2) A receiver that amplifies a restricted frequency band from the output of antenna.

(3) A recorder for registering the receiver output

Giant Meter-wave Radio Telescope (GMRT) Front Ends have been designed to operate at 5 frequency bands centered at 50 MHz, 150 MHz, 235 MHz, 327 MHz, 610 MHz and L-Band extending from 1000 to 1450 MHz. The L-Band is split into four sub-bands centered at 1060 MHz, 1170 MHz, 1280 MHz and 1390 MHz, each with a bandwidth of 120 MHz. The 150 MHz, 235 MHz and 327 MHz bands have about 40 MHz bandwidth and the 610 MHz band has about 60 MHz bandwidth. The low noise front end of the receiving system of GMRT has been designed to receive dual polarization. Lower frequency bands from 150 to 610 MHz have dual circular polarization channels (Right Hand Circular and Left Hand Circular polarization) which have been conveniently named as CH_1 and CH_2, respectively. The higher frequency L-Band has dual linear polarization channels (Vertical and Horizontal polarization) and they have been named CH_1 and CH_2 respectively. The front end system has flexibility to be configured for either dual polarization observation at a single frequency band or single polarization observation at two different frequency bands. The polarization channels can be swapped whenever required. For observing strong radio sources like "sun", the selectable solar attenuators of 14 dB, 33 dB can be used. The front end has RF termination facility also. Any band of the receiver can be switched OFF, whenever not in use, with the RF on/off facility provided in the front end. The receiver can be calibrated by injecting one of the four levels of calibrated noise, named Low cal, Medium cal, High cal and Extra-high cal depending upon the flux density of the source being observed. To minimize cross coupling between channels, a phase switching facility using walsh functions has been provided in the post unit. The functional schematic of the front end system is as shown in figure-1.2.

Figure -1.2: Functional schematic of the front end systems



The low noise amplifiers (LNA) for all the frequency bands are un-cooled. At 150 MHz band, Silicon Bipolar MMIC amplifier of Agilent Technologies has been used to design the LNA. The 235 MHz, 327 MHz and 610 MHz, the LNA designs are based on GaAs MESFETS from Agilent Technologies. For the L-Band LNA design, High Electron Mobility Transistors (HEMT) of Fujitsu have been used. For RF switching applications like band selection, RF on/off etc GaAs MESFET MMIC switches have been used. For polarizer designs, WIRELINE Quadrature hybrids of SAGE Laboratories make have been used for 150 MHz, 235 MHz and 327 MHZ bands. The 610 MHz band polarizer has been designed using Branchline Quadrature hybrid. Table -1.1 summarizes system noise temperatures for different frequency bands.

SYSTEM TEMPERATURES

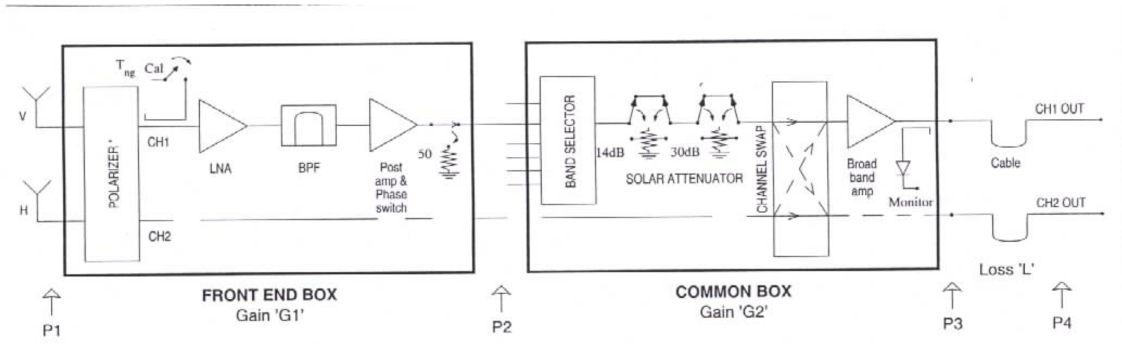
Frequency Band [MHz]	Input Cable Loss L' [dB]	Polarizer Loss L [K]	LNA Temp. T_{LNA} [K]	Receiver Temp. (Includes cable losses) T_R [K]	Ground Temp. T_{Gnd} [K]	Sky Temp. T_{Sky} [K]	System Temp. T_{Sys} [K]	Bandwidth [MHz]
150	0.2	0.75[1]	150	260	12	308	**580**	40
235	0.55[2]	0.25	35	103	32	99	**234**	40
327	0.13	0.18	30	55	13	40	**108**	40
610	0.22[3]	0.15	30	59	32	10	**101**	60
1060	0.22[4]	—	35	53	25	5	**83**	120
1170	0.22[4]	—	32	49	24	4	**77**	120
1280	0.22[4]	—	30	47	23	4	**74**	120
1390	0.22[4]	—	28	45	23	4	**72**	120

1. Includes 2–1 combiner insertion loss.
2. Insertion loss of Balun & associated cables.
3. Insertion loss of Balun & 20 cm 0.141" semirigid cable from Balun to Probe.
4. Contains loss of OMT & OMT to LNA input cable.

APK/GSS/TBA/ 25/11/99. tsys.fig

Table 1.1 – System Temperatures.

The RF power levels at various stages of the front ends are as shown in figure -1.3



FREQ. BAND (MHz)	T_{sys} (K)	P1 (dBm)	G1 (dB)	P2 (dBm)	G2 (dB)	P3 (dBm)	L (dB)	P4 (dBm)
150	580	- 96	34	- 62	28	- 34	5	-39
233	234	- 100	37	- 63	27	- 36	6	-42
327	108	- 103	38	- 65	27	- 38	7	-45
610	101	- 104	35	- 69	26	- 43	10	-53
1060	83	- 104	50	- 54	25	- 29	14	-43
1170	77	- 105	49	- 56	24	- 32	14	-46
1280	74	- 105	49	- 56	23	- 33	15	-48
1390	72	- 105	47	- 58	23	- 35	16	-51

Note : Power levels are over 32 MHz BW; with 0dB solar attenuator - * The L-Band (1000 - 1400 MHz) FE does not have the polarizer

Figure 1.3 - RF power levels at various stages of the front ends

1. 2 Receiver Parameters

1.2.1 Noise and Temperature

In general, noise is something indistinct, undesirable or even loud. In the world of electronics however, it is something that will always exist and must be dealt with. Even when an amplifier has no input signal, a small randomly fluctuating output signal can still be detected and is referred to as noise. Noise in electronic components is caused by random thermal fluctuations of electrons. Therefore any device operating above absolute zero will generate noise.

As commonly practiced in radio astronomy, noise is expressed in terms of noise temperature. The noise temperature of an antenna is defined as the thermal temperature of a resistor whose terminals, over a specified frequency bandwidth, have a mean available noise power due to thermal agitation equal to the available power at the antenna's terminals. Using temperature units allows direct comparison with celestial "source temperatures." Any object will emit electromagnetic waves if its temperature is above absolute zero. Ideally, the noise temperature of an antenna's radiation resistance will be equal to the temperature of the particular source the antenna is "looking at" if the angular extent of the source "fills" the antenna beam.

This statement excludes non-thermal mechanisms that generate electromagnetic waves such as synchrotron radiation. In such cases, the noise temperature of the antenna's radiation resistance will not equal the thermal temperature of the source but instead would be equivalent to the thermal temperature of an ideal blackbody emitting the same radiation at the observing frequency.

An ideal blackbody is defined as a perfect absorber and radiator: it absorbs radiation at all frequencies and its own radiation is a function of only temperature and frequency.

1.2.2 Doubling Sensitivity

Signals from distant celestial sources are weak to put things in perspective, the basic unit of power flux in radio astronomy is the Jansky. The Jansky is equivalent to only 10^{-26} W m^{-2} Hz^{-1}. It has been estimated that the energy from all the radio signals ever collected by all the radio telescopes on Earth amounts to no more than the kinetic energy of a falling snowflake. Receiver and antenna noise is an obstacle to detecting such small signals and must be minimized to maximize the sensitivity of a radio telescope. For the Synthesis Telescope, sensitivity in imaging can be thought of as the minimum detectable r.m.s. fluctuation ΔS whose units are W m^{-2} Hz^{-1}

1.2.3 Reducing Receiver Front-End Noise

By the time radio signals reach Earth from thousands of light years away, they are greatly attenuated and very weak. To detect these signals, receiver front-ends are required to have high gain and most importantly, very low noise. Noise and gain are important due to a particular property of amplifiers: first stage amplifier noise typically dominates the noise of the stages after it. For a chain of n amplifiers, the total noise temperature can be generalized

$$T = T_1 + [\, T_2 / G_1 \,] + [\, T_3 / (G_1 G_2) \,] + [\, T_4 / (G_1 G_2 G_3) \,] + \ldots \qquad \text{------------ (1.1)}$$

Where-

T = total noise temperature

Tn = noise temperature of nth amplifier

Gn = available power gain of nth amplifier

Therefore the noise performance of the first stage of the receiving system is the primary determining factor in the total noise performance of the system because it is only T1 that does not get reduced by power gain of other stages. A stage that contributes loss to the system and has physical temperature T attenuator will introduce a gain of less than unity as well as the following noise temperature T.

The key to obtaining low noise in an amplifier is the type of transistor that is used in the design. The most common types of transistors used in LNAs at microwave frequencies are gallium arsenide field-effect transistors (GaAs FETs) and high electron mobility transistors (HEMTs). Gallium arsenide is preferred over silicon due to its superior performance at microwave frequencies. GaAs FETs and HEMTs are further discussed in Chapter 2. Such new technology has made it possible for a 1420 MHz amplifier to have excellent noise performance without the need for cryogenic cooling. Cryogenic cooling of receivers is commonly used to achieve low system noise temperatures in radio astronomy. However, the aim of this project is to meet target performance of the Radio telescope without cooling. Although highly effective, cryogenic cooling is an expensive and high-maintenance method of reducing receiver noise.

1.3 Project goals (Objectives) :

Designing an LNA for lowest noise temperature requires knowing transistor noise parameters. Noise parameters can be obtained through a process that begins with measuring the S-Parameters of a FET and ends with using the circuit parameters of a FET model to calculate noise parameters. The final stage of LNA design involves building matching networks around the transistor which optimize the transistor for best noise performance. The main steps are summarized as following:

- S-Parameter Measurement
- Circuit Modeling
- Noise Parameter Calculation
- Noise Analysis
- Design of Matching Networks
- Stability Considerations
- Design of microstrip for 50 ohm characteristic impedance.

Since the Agilent transistor data-sheet S-parameters were not believed to be correct, it became necessary that S-parameters be measured for verification. S-parameters are useful because they directly reveal the impedance and gain of the transistor at a particular frequency. Furthermore, when obtained over a wide range of frequencies, S-parameters also provide a means to calculate the equivalent circuit model of a field-effect transistor (FET), which in turn allows calculation of transistor noise parameters. For an accurate circuit model, S-parameters needed to be measured over as wide a frequency range as possible. S-parameter measurement required the design and construction of test fixtures so that the transistors could be connected to a network analyzer for measurement.

CHAPTER 2

LOW NOISE AMPLIFIER (LNA) DESIGN CONCEPTS

In design of broad-band high frequency transistor amplifiers is a difficult and challenging problem in active network theory of great practical importance. The difficulties are largely due to the number of requirements at the same time. To set one of the parameter by changing the values of the appropriate circuit elements, another parameter gets affected. Therefore in broad-band high frequency transistor amplifiers, there is need of optimization of the component values to get optimum result. At present, a rigorous design theory for broad-band transistor amplifiers is not available and designers must rely on computer-aided and experimental cut and try techniques which are often inadequate.

Low noise microwave transistor amplifiers are primarily used in low noise receiving systems for radioastronomy, radar, satellite communications, terrestrial communications and allied purposes. The sensitivity of these receivers is often defined in terms of 'noise temperature', which is equivalent to noise figure. In the design of practical amplifiers, stability considerations is essential in addition to those for gain and noise. For low noise, design of impedance matching network is crucial part. Along with recent developments in the field of measurement instrumentation, the scattering matrix representation has become popular to characterize the microwave performance. Computer aided techniques are now widely used in the design of microwave circuits. The user should be aware of limitations of each technique for effective application.

Designing an LNA for lowest noise temperature requires knowing transistor noise parameters. Noise parameters can be obtained through a process that begins with measuring the S-Parameters of a FET and ends with using the circuit parameters of a FET model to calculate noise parameters. The final stage of LNA design involves building matching networks around the transistor which optimize the transistor for best noise performance. The main steps are summarized as following:

• S-Parameter Measurement
• Circuit Modelling
• Noise Parameter Calculation
• Noise Analysis and Design of Matching Networks

2.1 Design Considerations

While designing the LNA, we have to consider following required parameters of LNA

(1) Frequency range

(2) Noise temperature & Noise figure

(3) Gain (S-parameter)

(4) Input return loss (S-parameter) and Output return loss (S-parameter)

(5) 1 dB compression point

(6) 3^{rd} order intersect point

2.1.1 Frequency range

For GMRT radio telescope, the requirement of the frequency range is in between **500 MHz – 900 MHz** in P-Band (230 MHz – 1000 MHz – IEEE standard) which is also in Ultrahigh frequency range (UHF : 300 MHz - 3000 MHz)

2.1.2 Noise, Noise temperature & Noise figure

In a broad sense, noise can be characterized as any undesired signal that interferes with the main signal to be processed. Even when an amplifier has no input signal, a small randomly fluctuating output signal can still be detected and is referred to as noise. Noise in electronic components is caused by random thermal fluctuations of electrons. Therefore any device operating above absolute zero will generate noise. Noise arising in an electronic component or a device is historically classified as – (1) Thermal noise (2) Shot noise (3) Flicker noise (4) Partition noise

Thermal noise, also called Johnson noise occurs in any conductor due to random motion of electrons caused by thermal agitation. The shot noise is produced due to the random variations in the arrival of electrons at the output electrode of an amplifying device. For amplifying devices shot noise is inversely proportional to the transconductance of the device. In Unipolar devices like FET's, shot noise is so minimum that it can be neglected. Flicker noise is also called low frequency noise & appears at frequencies below few KHz. Flicker noise is generated due to fluctuations in the carrier density. These fluctuations in the carrier density will cause the fluctuations in the conductivity of the material. This will produce a fluctuating voltage drop when a direct current flows through a device. This fluctuating voltage is called as flicker noise voltage. Since flicker noise has a $1/f^{n}$ frequency spectrum with n close to unity, this noise can be neglected for microwave frequencies partition noise is is generated when the current gets divided between two or more paths. The devices like FETs, HEMTs draw almost zero gate bias current, hence keeping the partition noise to its minimum value.

As commonly practiced in radio astronomy, noise is expressed in terms of noise temperature. The noise temperature of an antenna is defined as the thermal temperature of a resistor whose terminals, over a specified frequency bandwidth, have a mean available noise power due to thermal agitation equal to the available power at the antenna's terminals. Using temperature units allows direct comparison with celestial 'source temperature'. Any object will emit electromagnetic waves if its temperature is above absolute zero. Ideally, the noise temperature of an antenna's radiation resistance will be equal to the temperature of the particular source the antenna is "looking at" if the angular extent of the source "fills" the antenna beam.

Noise and temperature are related in the following manner. For a resistor R at temperature T, the relation between noise power Pn and temperature T is expressed as follows

$$Pn = kT\Delta f \quad \text{---} \quad (2.1)$$

Where - Pn = noise power available from resistor (watts)
k = Boltzmann's constant (J/K)
T = temperature of resistor in Kelvin
Δf = operating bandwidth (Hz)

Ideally, above equation also applies to an antenna with radiation resistance R which is receiving signal from a blackbody at temperature T whose angular extent fills the antenna beam. Realistically however, the noise temperature referred to the antenna's terminals will consist not only of the source temperature but the noise temperature of the receiver it is connected to, not to mention additional noise from the thermal radiation of the ground (which gets "picked up" by antenna side lobes) and the sky. The total sum of noise temperatures can then be grouped into a single term: "system noise temperature."

Radio telescope system noise temperatures can vary from tens of degrees Kelvin to tens of thousands of degrees Kelvin depending on the frequency of operation. The radio frequency window itself extends from a few MHz to hundreds of GHz. The lower system temperatures apply to microwave (centimeter wavelength) frequencies.

Noise figure F of the two-port network is the ratio of signal-to-noise power ratio at the input to signal-to-noise ratio at the output. In other words, noise figure is a measure of how much a signal degrades as it passes through the two-port as a result of the two-port's own noise. An expression for the noise figure of the two-port in terms of input noise temperature T_s and additional effective input noise temperature T_e:

$F = (P_{Si} / P_{Ni}) / (P_{So} / P_{No}) = 1 + T_e / T_s$ -------------------------------------- (2.2)

Where – P_{si} – Input signal power

P_{So} - Output signal power

P_{Ni} - Input noise power

P_{No} - Output noise power

T_e - Noise temperature of the noisy two-port.

T_s - Reference input noise temperature required to obtain the noise figure

T_e has already been defined as the noise temperature of the noisy two-port. T_s, also known as T_0, is a reference input noise temperature required to obtain the noise figure of the two-port. The standard value has been set at 290 K. The term"noise figure" has been commonly used since the 1940's, when it was defined as above by Harold Friis. In modern usage, however, the quantity F has often come to be known as "noise factor," while the same quantity in dB units, NF, is known as noise figure.

NF = 10 log F. -- (2.3)

For simplicity however, in this thesis F and NF will both be referred to as noise figure — NF being noise figure in terms of dB.

ATF-54143 demonstration amplifier has a nominal 0.5 dB (refer : data sheet) noise figure.

2.1.3 Gain (S-parameter)

S-parameters are a valuable aid both for collecting data for a transistor and then using the data to predict performance and design an amplifier circuit. *The values of S parameters depend not only upon the properties of the transistor but also upon the source and load circuits used to measure them.* This is because they measure transmitted and the source and load used to test it.

The general transducer gain (G_T) of a two port network having S_{21} & S_{12} values, whether it is a transistor or not, is

$G_T =$ Power delivered to load / Power available from source

In terms of s-parameters, transducer gain (G_T) is given by,

$$G_T = [(1- | \Gamma s |^2) / (| 1- S_{11} \Gamma s |^2)] [| S_{21} |^2] [(1- | \Gamma_L |^2) / (| 1- S_{22} \Gamma_L |^2) \quad \text{----------------------- (2.4)}$$

The transducer gain expressions are too complex for manual design. To effect an approximate gain solution, let us ignore the feedback, that is assume that $S_{12} = 0$. If an amplifier has no feedback, signals pass one way through it. Accordingly this is called unilateral gain.

If $S_{12} = 0$, then $\Gamma_{IN} = S_{11}$ and $\Gamma_{OUT} = S_{22}$ and transducer gain becomes the unilateral gain.

The Overall gain (in dB) of the transistor amplifier is given by,

$$G_{TU}(dB) = G_S(dB) + G_O(dB) + G_L(dB) \quad \text{-- (2.5)}$$

Where - G_S and G_L are the gains to be obtained by matching the input and output circuits, respectively. If in the actual transistor, $S_{12} \neq 0$. In that case the true gain G_T is related to the calculated unilateral gain G_{TU} by,

$$[1/ (1 + U)^2] < [G_T / G_{TU}] < [1/ (1 - U)^2] \quad \text{------------------------------------ (2.6)}$$

$$\text{Where- } U = [| S_{11} | | S_{21} | | S_{12} | | S_{22} |] / [(1-| S_{11} |^2) (1-| S_{22} |^2)] \quad \text{----------------(2.7)}$$

Gain of at least 33 dB or greater is required for a GMRT Radio Telescope front-end LNA so that the noise of the subsequent stages will be sufficiently reduced by the gain of the first stage. Because the ATF- 54143 is a potentially unstable transistor, the value for maximum stable gain should be used as a figure of merit. Maximum stable gain depends on the forward and reverse transmission S-parameters

$G_{MSG} = |S_{21}| / |S_{12}|$; Where G_{MSG} - is maximum stable gain. ---------------------- (2.8)

In the case of the amplifier using ATF-54143, the maximum stable gain at 700 GHz is 16.6 dB (refer : data sheet). Therefore, an LNA requires at least two transistor stages for sufficient gain as a GMRT Radio Telescope front-end requires.

2.1.4 Input return loss (S-parameter) and Output return loss (S-parameter)

Input return loss indicates the mismatch between the input impedance of an amplifier and the characteristic impedance of the transmission line at its input by acting as a measure of the amount of power reflected back when an input signal is injected into the input of the amplifier through the transmission line. The input return loss value can be expressed in terms of the measured S parameter S_{11}:

Input Return Loss = $-S_{11}(\text{dB}) = -20 \log |S_{11}|$. -------------------------------------- (2.9)

Similarly, output return loss is represented by S-parameter S_{22}:

Output Return Loss = $-S_{22}(\text{dB}) = -20 \log |S_{22}|$. ----------------------------------- (2.10)

Input return loss is of more concern than output return loss, however, because too big a mismatch at the input of the LNA can lead to degradation in noise performance. During the course of this project, commercial amplifiers were purchased and measured with the N8973A noise meter to have excellent noise temperatures of under 20 K (< 0.3 dB). However, once the commercial amplifiers were installed on one of the GMRT Telescope dishes for testing, system noise temperatures were considerably higher than anticipated, and some amplifiers were unstable when connected to the waveguide feed. These problems may be attributed to the extremely poor input return losses of the amplifiers: measured input return loss for the commercial amplifiers averaged only 0.5 dB, which is equivalent to 89% of the input power being reflected back from the input of the LNA.

When there is absolutely no mismatch between the input of the LNA and the transmission line, no reflections occur and the input return loss is infinite. Maximum power transfer from the source to the input of the LNA is achieved. In actuality there will always be some loss, even input return losses on the order of 5 dB (about 70% of the input power is absorbed by the input of the LNA) are acceptable. Maximum power transfer occurs when the impedance of the power source and the impedance of the load are conjugates of each other. In general, the source impedance is equal to the standard value for coaxial transmission line characteristic impedance: 50 Ω . Therefore maximizing transmission of signal from the source to the LNA input requires building an input matching network that transforms 50 Ω to the conjugate of the input impedance of the transistor, $Z^{*}in$. One cannot design for maximum power transfer and lowest noise simultaneously because generally $Zopt \neq Zin^{*}$ and designing for lowest noise involves building an input matching network that transforms the source impedance of 50 Ω to $Z_s = Zopt$ instead of $Z_s = Zin$.

A way to improve the input return loss of an LNA while maintaining good noise performance consists of adding extra series inductance at the source terminal of the transistor. Inductance L_s at the source terminal provides feedback to the input impedance of the transistor by adding ($g_m.Ls / Cgs$) to the real part of the intrinsic circuit's Zin and adding ($\omega Ls - 1 / \omega\ Cgs$) to the imaginary part of Zin without changing Z_{opt} significantly. On the Smith Chart, $Z_s = Zin$ moves closer to $Z_s = Zopt$ with increasing source inductance. There is a limit to the amount of source inductance that can be added. Excessive source inductance can lead to LNA oscillations because of gain peaks at higher frequencies. For the ATF-54143, source inductances on the order of a few tenths of nH to a few nH are sufficient.

2.1.5 1 dB compression point :

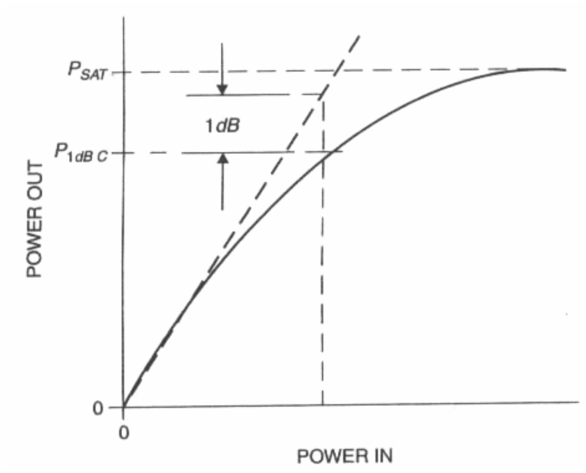


Figure 2.1 - 1 dB compression point

An amplifier is called linear when the output power increases linearly with the input power. The ratio of these two powers is the power gain G. as input power increases, the amplifier transfer function becomes nonlinear, that is the output power is lower than predicted by the small signal gain. This nonlinear behavior in amplifiers introduces distortion in the amplified signal. The output power at which the gain has dropped by 1 dB below the linear gain is called the 1 dB compression point, P_{1dB}. Typically the gain will drop rapidly for powers above 1 dB, reaching a maximum or fully saturated output power within 3 to 4 decibels above P_{1dB}. There are a number of different ways to measure the nonlinearity behavior of an amplifier. The simplest method is the measurement of 1 dB compression power level P_{1dB}.

1 dB compression point, P_{1dB}, is defined as the output power at which the output power of the network is 1 dB less than it would have been had its input to output characteristic remained linear. Beyond the 1 dB compression point, as input power to the amplifier is further increased, the output power reches a maximum saturated power output, P_{SAT}.

2.1.6 Third order intersect point :

It is the another method to measure the nonlinearity behavior of an amplifier. This popular method uses two closely spaced signals 5 to 10 MHz apart. When two signals at frequencies, f_1 and f_2 are incident on an amplifier, the output of the amplifier contains these two signals, as well as inter-modulation products at frequencies $m f_1 + n f_2$, where m + n is known as order of inter-modulation (IM) product.

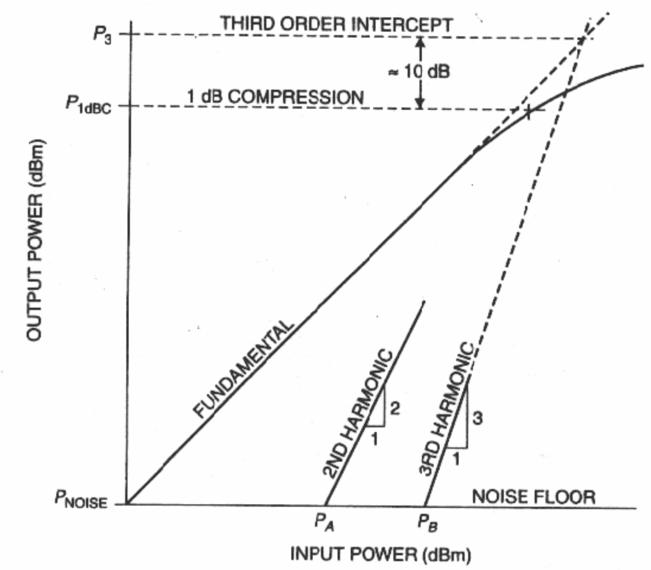


Figure 2.2 - Third order intersect point

The relative magnitude of the inter-modulation products depends on the details of how the amplifier saturates, however, third order products which are closest in frequency to the fundamental tones usually dominate at moderate saturation levels, although second order products can be important in multioctave amplifiers.

With proper filtering, one of the fundamental frequencies and the distortion products are measured separately. Since second & third order IM products correspond to square & cubic nonlinearities respectively, the output power for these products increases by 2 dB/dB and 3 dB/dB at low power levels. Since the third order terms normally dominate up to the point where distortion is very severe, intermodulation distortion is often characterized by the third order intercept, P_{3rd} . Intercept P_{3rd} is the point where the power in the third order product and fundamental tones are equal when the amplifier is assumed to be linear. The third order intercept power is typically 10 to 12 dB above the P_{1dB} and is a very useful parameter for calculating low level inter modulation effects.

2.2 Field Effect Transistor Family

The transistor is traditionally classified into two groups, namely, bipolar and unipolar. The bipolar junction transistor (BJT) is based on the phenomenon of minority-carrier injection into a region where majority carriers are present. The injected carriers move primarily through diffusion process. In the unipolar group, practically, all of the current flowing through a device is due to majority carriers.

The field effect transistors belongs to this family, in which the conductivity of a conducting channel is modulated by a transverse electric field under the controlling electrode. In BJT, full shot noise is associated with each current flowing in the device. This noise is caused due to random processes of diffusion and/or recombination. In FETs the noise voltage distribution occurs along the channel due to distributed thermal noise sources. At microwave frequencies this noise voltage gives rise to a noise current flowing to the gate by capacitive coupling. This induced gate noise is partially correlated with the drain noise.

A simple shot noise picture can give a reasonable description of most noise phenomenon in BJTs, whereas a simple thermal noise model can explain most noise behaviors of FETs. This is the key distinction between noise properties of BJTs and FETs. In recent years, the noise performance of a microwave transistor has been remarkably improved.

Unlike BJTs, FETs are unipolar devices, meaning that only one carrier type, either holes or electrons, contributes to the current flow through the channel. If hole contributes are involved, it is called p-channel, Otherwise n-cannel FETs. Moreover, the FET is a voltage controlled device.

Traditionally FETs are classified according to how the gate is connected to the conducting channel. Specifically the following four types are used :

(1) **Metal Insulator Semiconductor FET (MISFET)** : Here the gate is separated from the channel through an insulation layer. One of the most widely used types , Metal Oxide Semiconductor FET (**MOSFET**), belongs to this class.
(2) **Junction FET (JFET) :** This type relies on a reverse biased pn-junction that isolates the gate from the channel.
(3) **MEtal Semiconductor FET (MESFET) :** If the reverse biased pn junction is replaced by a Schottky diode, the channel can be controlled just as in the JFET case.
(4) **Hetero FET :** As the name implies, the hetero structures utilize abrupt transitions between layers of different semiconductor materials. Ex. GaAlAs. The High Electron Mobility Transistors (HEMT) belongs to this class.

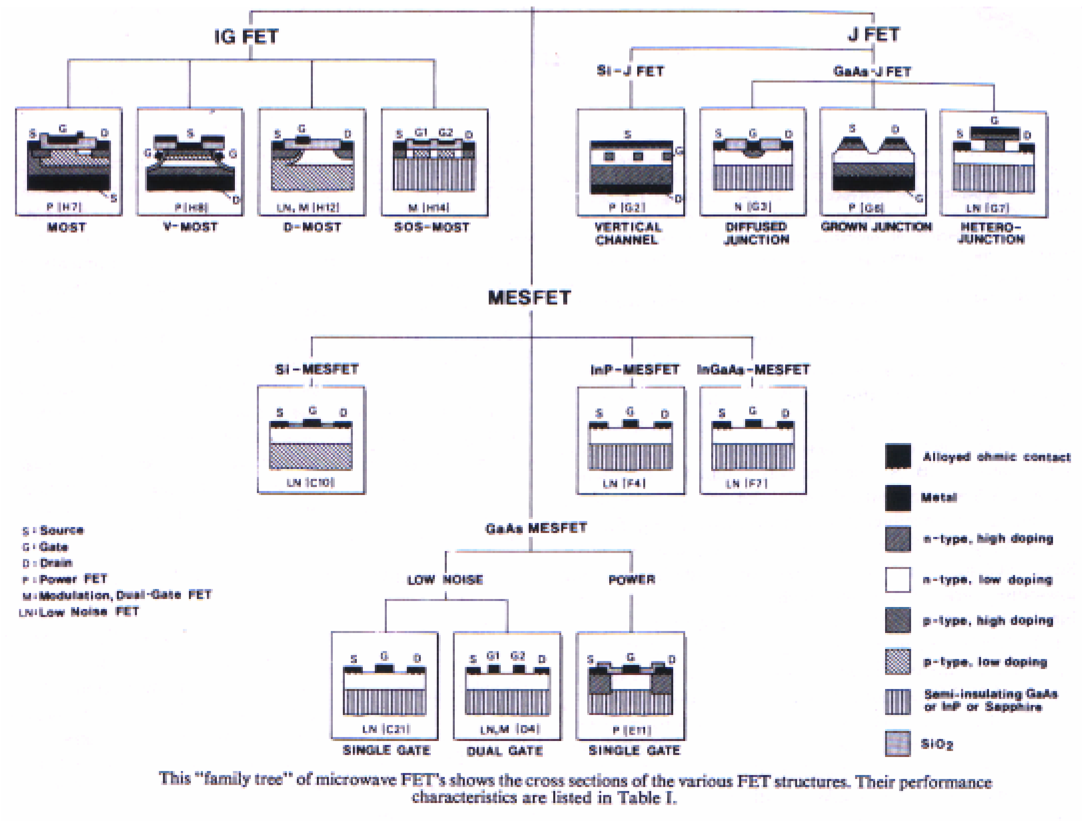
FETs are electrically classified according to **enhancement** and **depletion** types. This means that the channel either experiences an increases in carriers (for instance the n-type channel is injected with electrons) or a depletion in carriers (for instance the n-type channel is depleted of electrons)

High Electron Mobility Transistors :

The high electron mobility transistor (**HEMT**), also known as modulation-doped FET (**MODFET**) exploits the difference in band gap energy between dissimilar semiconductor materials such as GaAlAs and GaAs in an effort to substantially surpass the upper frequency limit of MESFET while maintaining low noise performance and high power rating. At present, transit frequencies of 100 GHz and above have been achieved. The high frequency behavior is due to a separation of the carrier electrons from their donor sites at the interface between the doped GaAlAs and undoped GaAs layer, where they are confined to a very narrow layer in which motion is possible only parallel to the interface. Here we speak of a two-dimensional electron gas or plasma of very high mobility.

HEMTs are primarily constructed of heterostructures with matching lattice constants to avoid mechanical tensions between layers. Specific examples are the GaAlAs-GaAs and InGaAs-InP interfaces. Research is also ongoing with mismatched lattices whereby, for instance, a larger InGaAs lattice is compressed onto a smaller GaAs lattice. Such device configurations are known as **pseudomorphic HEMTs** or **pHEMTs**

Following figure 2.3 shows the ‘family tree’ of microwave FETs and cross-section of various FET structures.



This "family tree" of microwave FET's shows the cross sections of the various FET structures. Their performance characteristics are listed in Table I.

Figure 2.3 - FET family tree.

2.3 How Field Effect Transistors (FETs)Work

The FET is a three-terminal semiconductor device, represented by figure 2.4. The voltage between two terminals of a FET controls the current flow in the third terminal, enabling the device to function as an amplifier. These three terminals are referred to as source, gate and drain. It is the voltage between gate and source that controls the current in the drain.

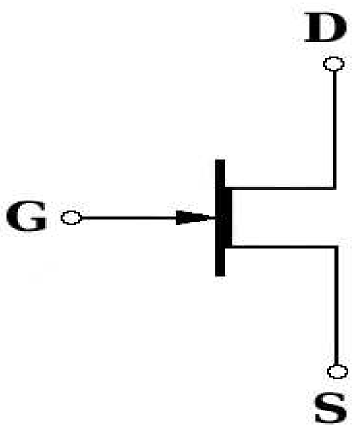


Fig. 2.4 - Symbolic representation of FET

The late 1970's brought into existence different types of FETs starting with the MOSFET (metal-oxide semiconductor FET). The MOSFET as well as numerous other types of FETs are silicon-based. Of interest here, however, are the more newly developed gallium arsenide (GaAs) based devices. GaAs FETs are also commonly referred to as MESFETs (metal-semiconductor FETs). MESFETs became commercially available in the 1980's. At higher frequencies (on the order of GHz), gallium arsenide has superior performance compared with that of silicon, due to the high electron drift mobility of GaAs, which is five to ten times higher than that of silicon . At the time work began on this thesis, there were two main types of GaAs devices available: MESFETs and HEMTs (high electron mobility transistors). HEMTs were even more recently developed than MESFETs and have come to exceed MESFETs in noise performance.

2.4 Metal-Semiconductor Field Effect Transistors (MESFETs)

A MESFET is fabricated on undoped GaAs substrate (undoped GaAs has very low conductivity and isolates devices on a chip from each other as well as ensuring small capacitances between the devices and ground). The source and drain terminals are heavily n-doped GaAs regions with metal contacts. The gate terminal, however, consists of a Schottky-barrier junction, which is a metal-semiconductor junction that behaves like a diode (conducts current when voltage is applied in a particular direction and acts like an open circuit when the voltage is reversed). Current flows from drain to source by means of a conducting channel consisting of n-type GaAs which is not as heavily doped as the source and drain contact regions. Applying a small negative gate voltage creates an electron depletion region in the channel. The thickness of the depletion region (which in turn determines the physical dimensions of the channel) is determined by the value of the voltage applied, thus allowing the gate voltage to control drain-to-source current. Channel length is defined by the length of the gate electrode and is typically 0.2-2 μm. Channel width (the dimension of the gate electrode into the page) is determined by the designer. Refer to Figure 2.5 for a typical MESFET cross-section.

.

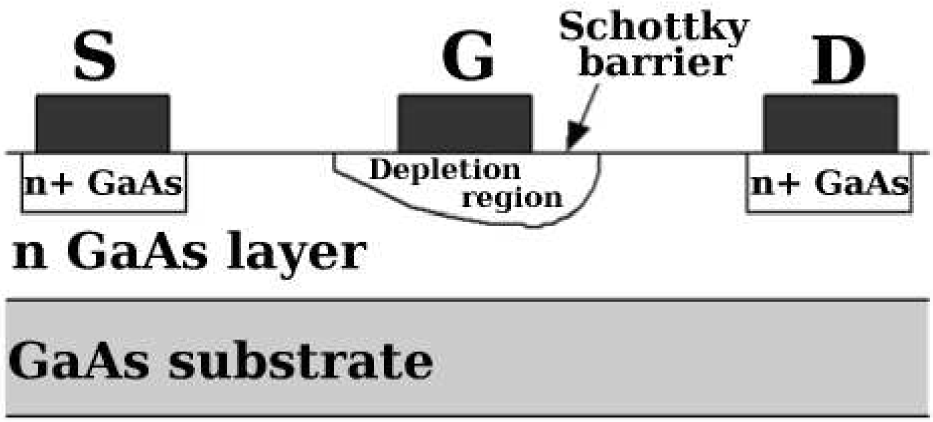


Fig. 2.5 - MESFET Cross-section

2.5 High Electron Mobility Transistors (HEMTs)

A HEMT can be thought of as a MESFET with a heterojunction. A heterojunction is the boundary between two layers composed of materials with different band gaps, a band gap being the energy difference between the valence band and the conduction band of a semiconductor. The layers are commonly composed of GaAs and n-type gallium aluminum arsenide (AlGaAs) as shown in Figure. The eterojunction provides a very low resistance channel almost free of impurities for electron carriers. This high electron mobility channel is also commonly referred to as a 2D (two-dimensional) electron gas. The high mobility and the freedom from impurities allow electrons to move with relatively few lattice collisions, with the result that noise generated in the transistor is very low. Figure 2.6 shows HEMT cross-section.

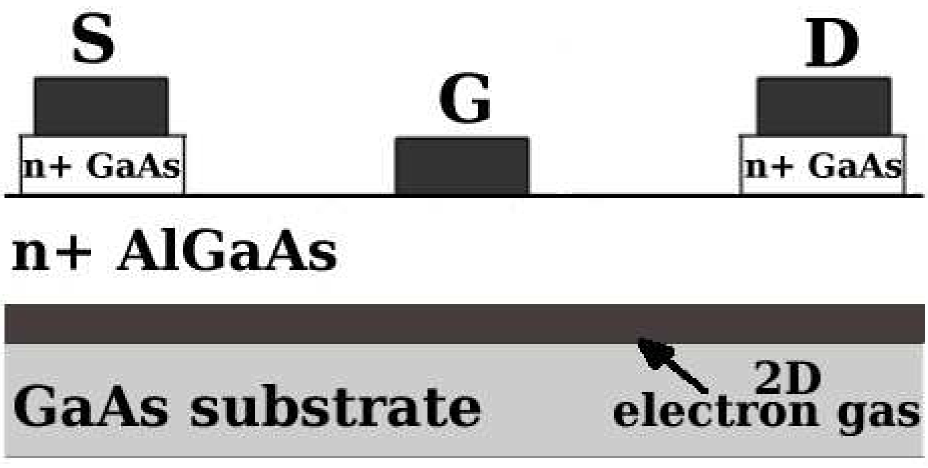


Fig. 2.6 HEMT Cross-section

2.6 Pseudomorphic High Electron Mobility Transistors (pHEMTs)

The transistor chosen for the project is an Agilent pHEMT (pseudomorphic HEMT), shown in Figure 2.4. Essentially, pHEMTs are HEMTs in which different layers have not only different band gaps but different lattice constants (spacing between the atoms) as well. A greater number of different types of layers to choose from allows for an even bigger bandgap difference between layers, further improving electron mobility in the channel. pHEMTs are currently the fastest operating transistors available - the frequency of operation reaches hundreds of GHz. Of interest here, however, is noise performance at 0.610 GHz. At the time commercial transistors were being investigated for this project, the Agilent ATF-54143 pHEMT became the prime candidate due primarily to its noise temperature. Agilent specifications state a noise temperature of 36 K when operated at 2 GHz for the ATF-54143 when DC biased at 4 V and 60 mA, which was one of the best commercial transistor noise performances at the time of investigation. FET gate width is determined by the designer: the ATF-54143 pHEMT has a gate width of 800 μm. The wider gate width of the ATF-54143 provides impedances that are easier to match to as well as high linearity. Linearity is important in amplifiers: non-linearity leads to intermodulation, the generation of in-band signals by out-of-band interference.

2.7 Selection of ATF 54143 pHEMT for LNA design

We have selected ATF 54143 pHEMT to design LNA .Agilent Technologies's ATF-54143 is a high dynamic range, low noise, E-PHEMT housed in a 4-lead SC-70 (SOT-343) surface mount plastic package. The combination of high gain, high linearity and low noise makes the ATF-54143 ideal for low noise amplifier in the 450 MHz to 6 GHz frequency range. This transistor satisfies the requirement of GMRT LNA. Hence it is selected.

Features

• High linearity performance

• Enhancement Mode Technology

• Low noise figure

• Excellent uniformity in product

• 800 micron gate width

• Low cost surface mount small plastic package SOT-343 (4 lead SC-70)

• Tape-and-Reel packaging option available

Specifications

• 2 GHz; 3 V, 60 mA (Typ.)

• 36.2 dBm output 3rd order intercept

• 20.4 dBm output power at 1 dB gain compression

• 0.5 dB noise figure

• 16.6 dB associated gain

Surface Mount Package SOT-343

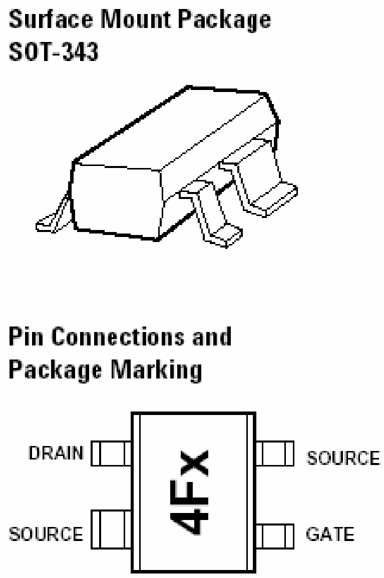


Figure 2.7 – ATF 54143 pin configuration and package

As specifications of ATF 54143 pHEMT fulfills the design requirement of LNA, we have selected it.

2.8 S-Parameters and the Smith Chart

At microwave frequencies, wavelengths become comparable in size to the circuit. In such a case, simple circuit analysis does not suffice because the phase of electrical signals will change significantly as the signals travel the length of the circuit. Instead, voltages have to be modelled for circuit analysis. As shown in Figure 2.8, the most basic picture consists of a voltage wave leaving source impedance Z_S, traveling along a transmission line with characteristic impedance Z_O, and then reaching load impedance Z_L. If the load impedance is equal to the characteristic impedance of the transmission line ($Z_L = Z_O$), the wave is fully absorbed by the load and none of it is reflected back towards the source Z_S. If there is a mismatch between the load impedance and the characteristic impedance of the line (Z_L = ZO), a portion of the wave is reflected back towards the source Z_S.

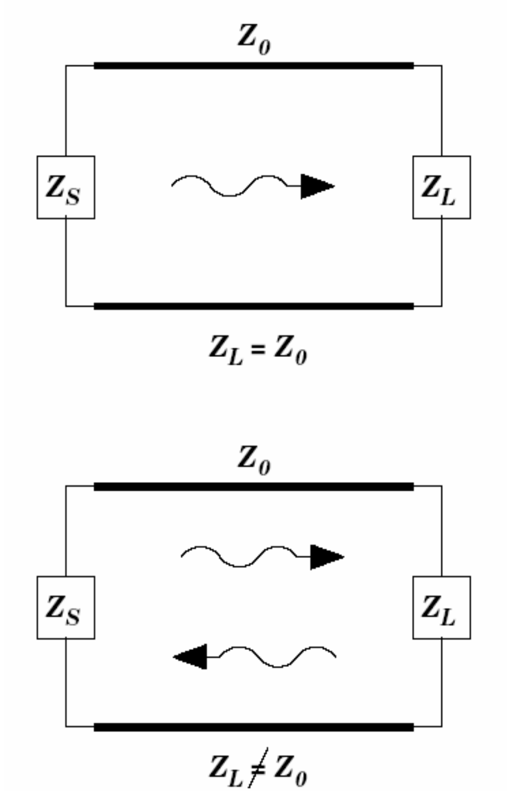


Fig 2.8 : Dependence of reflection & transmission on Z_L & Z_0

Terminating a port with a load equivalent to the characteristic impedance of the transmission line eliminates any reflected waves due to mismatch in transmission line and load. Characteristic impedance of a transmission line can therefore be defined as the value of impedance that, when attached to the end of a line, eliminates any reflection. A real value of 50 Ohms is the industry standard value for characteristic impedance..

To represent a two-port network at microwave frequencies, scattering parameters (S-parameters) can be used. S-parameters themselves (S_{11}, S_{12}, S_{21}, S_{22}) represent reflection and transmission coefficients of the two-port under certain “matched” conditions. S_{11} is the reflection coefficient and S_{21} is the transmission coefficient at port 1 when port 2 is terminated in a load whose impedance is equal to that of the transmission line characteristic impedance. Likewise S_{22} is the reflection coefficient and S_{12} is the transmission coefficient at port 2 when port 1 is terminated in a matched load.

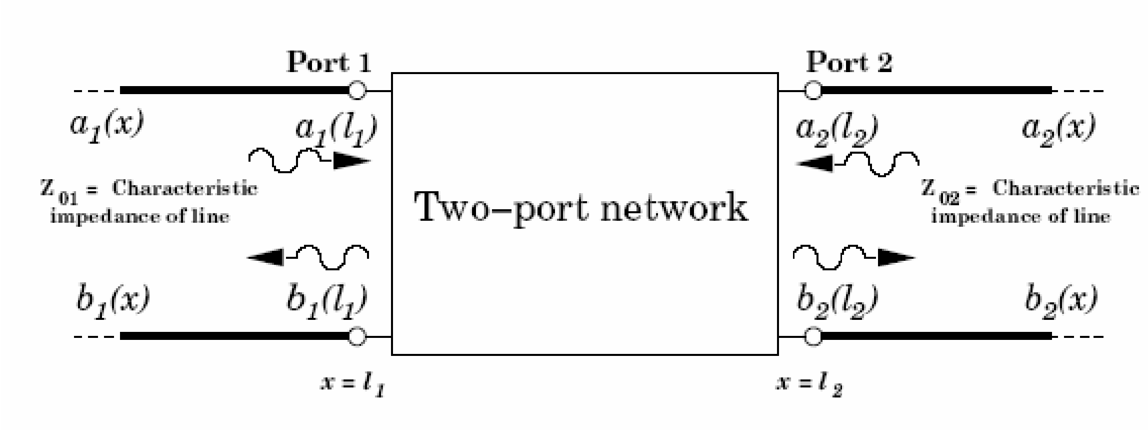


Figure 2.9: Incident and reflected waves in a two-port network

In matrix form,

$$\begin{bmatrix} b_1(l_1) \\ b_2(l_2) \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{bmatrix} a_1(l_1) \\ a_2(l_2) \end{bmatrix}$$

where $a1(\ell 1)$and $a_2(\ell 2)$ represent incident waves and $b_1(\ell 1)$ and $b_2(\ell 2)$ represent reflected waves. Note that when the output port is terminated with -

$Z_L = ZO2$, $a2(\ell 2) = 0$ (no reflection occurs at Z_L). Therefore S_{11} and S_{21} can be defined as the following:

$S_{11} = b1(\ell 1) / a1(\ell 1)$ ____$a2(\ell 2) = 0$ (input reflection coefficient; output port matched)

$S_{21} = b2(\ell 2) / a1(\ell 1)$ ____$a2(\ell 2) = 0$ (forward transmission coefficient; output port matched)

Similarly, when the input is terminated with $Z_S = Z_{O1}$, $a1(\ell 1) = 0$ (no reflection occurs at Z_S) and S_{22} and S_{12} are defined as follows:

$S_{22} = b2(\ell 2) / a2(\ell 2)$ ____ $a1(\ell 1) = 0$ (input reflection coefficient; input port matched)

$S_{12} = b1(\ell 1)/a2(\ell 2)$____$a1(\ell 1) = 0$ (reverse transmission coefficient; input port matched)

2.9 Reflection Coefficients and Impedance

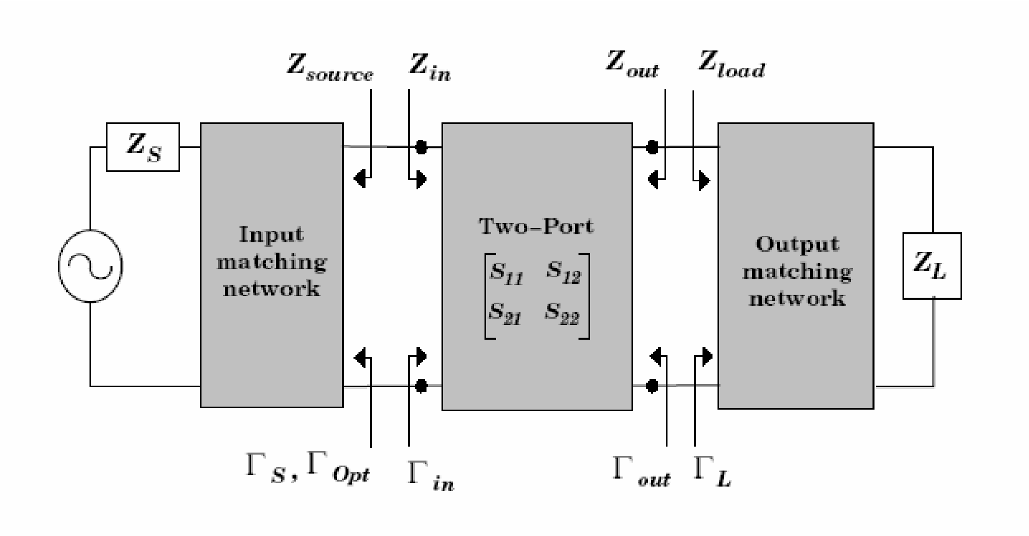


Figure 2.10: Generalized analytical form of the low noise amplifier.

At a particular node, the voltage reflection coefficient Γ is related to the complex impedance Z seen at that point by means of the following equation where Z_0 is the characteristic impedance of the line (usually a real value of 50)

$\Gamma = (Z - Z_0) / (Z + Z_0)$ --- (2.11)

Inversely,

$Z = Z_0 (1 + \Gamma) / (1 - \Gamma)$ --- (2.12)

Associated with the two-port are input and output reflection coefficients Γ_{IN} and Γ_{OUT} . Source and load reflection coefficients Γ_S and Γ_L are respectively associated with input and output matching networks as shown in the following.

When the two-port is a transistor, its performance is optimized by inserting a matching network at each port. The input matching network presents reflection coefficient Γ_S at the "source" side of the two-port and the output matching network presents Γ_L at the "load" side. Each matching network is designed so that each port sees a desired impedance. For example, when designing for maximum power transfer, each port should"see" its own impedance conjugate. With reference to Figure, Z_S should see Z^*_S when looking at the input matching network. In turn, the two-port should see Γ^*_{in} when looking at the input matching network. Similarly at the output of the two-port, Z_L should see Z_L when looking at the output matching network and $\Gamma_L = \Gamma^*_{out}$. Designing for minimum noise however, has only one requirement: the input port of the transistor needs to see the optimum reflection coefficient. To achieve minimum noise, $\Gamma_S = \Gamma_{opt}$.

2.10 The Smith Chart

The Smith Chart is a straightforward way of displaying reflection coefficients and corresponding impedances at the same time. The Smith Chart is a plot of impedances (and/or admittances) mapped onto the Γ -plane. Since the Smith Chart is in Γ -plane format, Γ can be plotted in polar form. | Γ | = 1 implies 100% reflection or fully reflected voltage waves while | Γ | = 0 (the center of the Smith Chart) implies zero reflection and fully transmitted waves. Each polar point on the Smith Chart corresponds to a particular impedance.

On the Smith Chart, impedance is normalized by the characteristic impedance value Z_o (usually a real value of 50):

$$z = Z / Z_o = (R + jX) / Z_o = r + jx \quad \text{--} \quad (2.13)$$

z = normalized impedance
Z_o = characteristic impedance
Z = impedance
R = resistance (real part of impedance)
X = reactance (imaginary part of impedance)
r = normalized resistance
x = normalized reactance

The relationship between z and Γ forms constant resistance r circles on the Smith Chart as shown in Figure 2.11.

$$z = Z / Z_o = (1 + \Gamma) / (1 - \Gamma) = (R + jX) / Z_o = r + jx \quad \text{---------------------------} \quad (2.14)$$

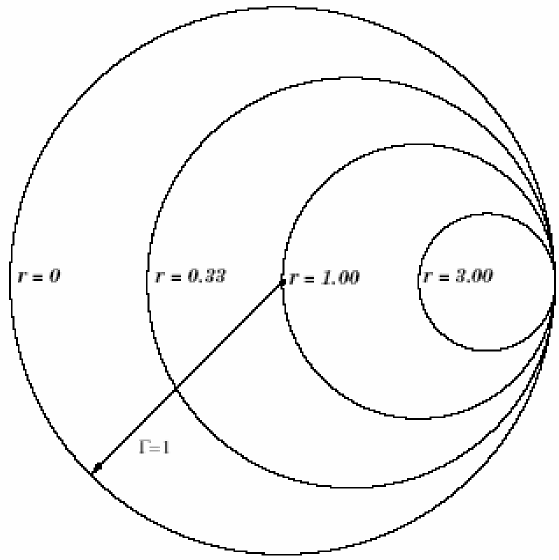


Figure 2.11- Normalized resistance circles on the Smith Chart.

Normalized reactance x forms curves on the Smith Chart which are orthogonal to the resistance circles as shown in Figure 2.12

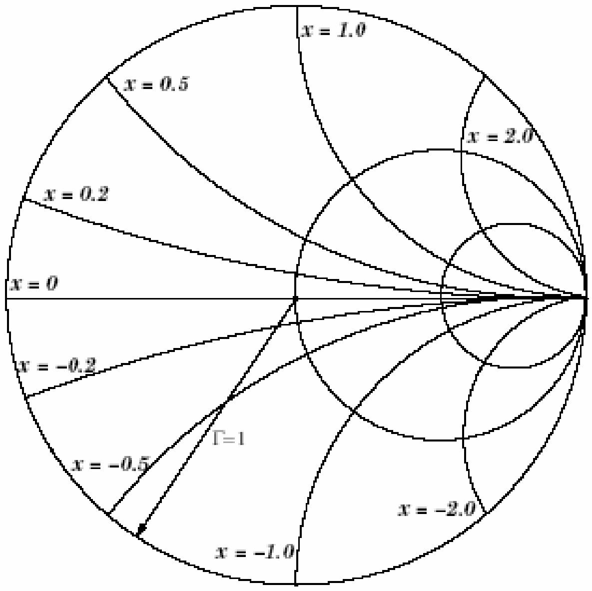


Figure 2.12 - Normalized reactance curves on the Smith Chart.

Similarly for normalized admittance y,

$$y = Y Z_o = (G + jB)Z_o = g + jb \quad \text{--} \quad (2.15)$$

where-

y = normalized admittance

Z_o = characteristic impedance

Y = admittance

G=conductance (real part of admittance)

B = susceptance (imaginary part of admittance)

g = normalized conductance

b = normalized susceptance

In accordance with r and x respectively, g forms constant conductance circles on the Smith Chart while b forms constant susceptance curves

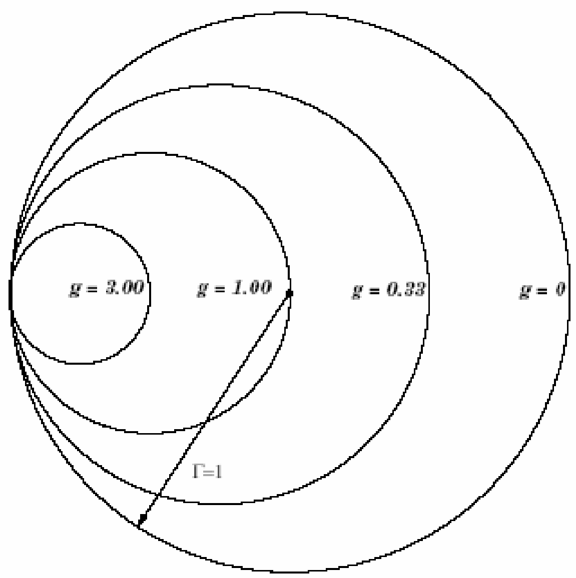


Figure 2.13 - Normalized conductance circles on the Smith Chart.

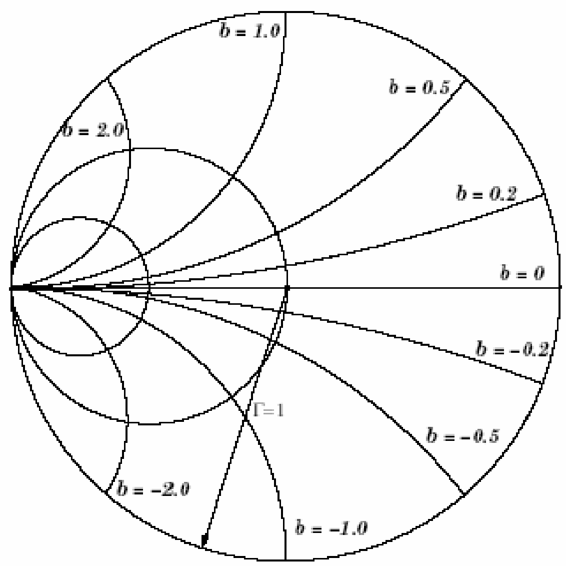


Figure 2.14 - Normalized susceptance circles on the Smith Chart.

The circumference of the Smith Chart is marked with degrees and corresponding fractions of a wavelength. Rotation along a constant radius in the Smith Chart corresponds to motion along the transmission line, where the center of the circular motion corresponds to the normalized characteristic impedance of the line. Clockwise motion corresponds to motion away from the source and towards the load. One full 360^0 rotation around the Smith Chart is equivalent to a displacement of half a wavelength along the transmission line. This is because we are measuring reflection coefficients, which means the path of a traveling wave includes "forward" and "backward" motion. If the point of reflection is half a wavelength away, the wave would have still undergone a 360^0 phase shift going there and then back to the point of measurement.

Once understood however, the Smith Chart becomes a useful visual tool in designing matching networks

2.11 Transistor S-Parameter Measurement :

Transistor S-parameters are an essential part of the transistor model used in amplifier design. Even if an amplifier is being designed for a single particular frequency, acquiring transistor S-parameters over a large range of frequencies is valuable for modelling purposes. Using high frequency circuit simulation software, the circuit model of a transistor can be calculated by fitting simulated S-parameters to measured S-parameters. A larger number of frequency points results in a more accurate transistor model.

Because wavelengths are short at high frequencies, transistors must be very small. Coupling transistors to laboratory instruments becomes very difficult, and test fixtures are needed to provide the interface. S-parameters of the Agilent ATF-54143 pHEMT are measured with a network analyzer using test fixtures. Thru-Reflect-Line (TRL) calibration, a method of in-fixture calibration, was performed in order to remove the effects of the test fixtures and obtain the S-parameters of the ATF-54143.

2.12 FET Modelling and Noise Parameter Calculation

The noise parameters of a FET can be calculated upon obtaining its equivalent circuit noise model. Theoretically, the noise parameters depend on certain parameters of the circuit model. The circuit model of the transistor is obtained by iteratively fitting the model S-parameters to the actual measured S-parameters of the device. There are various methods of doing this. Here, the chosen method is centered around least-squares fitting of the simulated output of a circuit model to measured S-parameters. After convergence, reliable values for various circuit elements such as package capacitances and inductances are obtained. The procedure used for calculating noise parameters is summarized in Figure

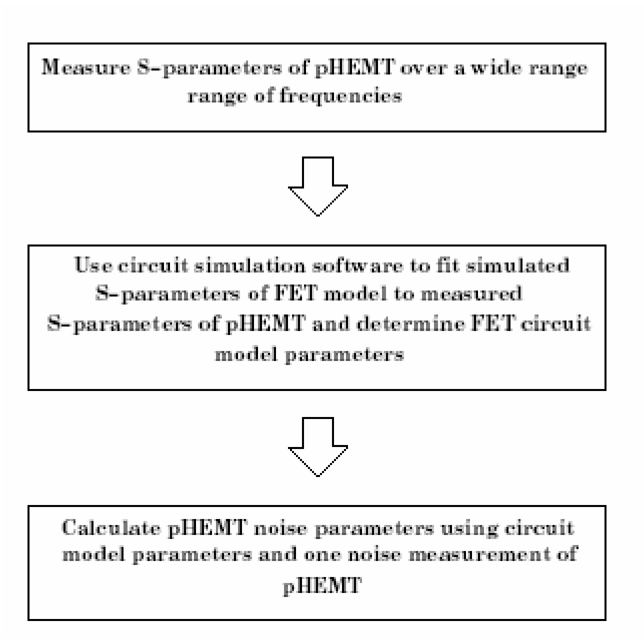


Figure 2.15 - Procedure for calculating pHEMT noise parameters.

2.13 Noise Analysis and Design

After the FET modelling described in the previous section, a good knowledge of S-parameters and the noise parameters can be obtained. There are four noise parameters:

- T_{min}
- the real part of Z_{opt}
- the imaginary part of Z_{opt}
- R_n

The most important are T_{min} and the real and imaginary parts of Z_{opt} . T_{min} is a value that specifies the lowest noise temperature the FET can have. Z_{opt} is the impedance the FET's input should "see" in order to achieve T_{min} (recall for minimum noise $\Gamma_S = \Gamma_{opt}$). The fourth parameter R_n is referred to as the equivalent noise resistance and is a sensitivity parameter for mismatches between Z_{opt} and Z_S.

Knowledge of a FET's S-parameters and noise parameters allows for the design of the required input matching network in the LNA circuit. Once Z_{opt} is obtained, an input matching network can be designed to present the desired impedance to the transistor input. Noise analysis and the design process are further documented in respective Chapters.

.

CHAPTER 3

MEASUREMENT OF TRANSISTOR S-PARAMETERS & NOISE PARAMETERS

Successful S-parameter measurement of the Agilent ATF-54143 pHEMT was achieved up to 10 GHz using TRL calibration. This chapter covers the method of calibration used as well as the components used to construct the test and calibration fixtures for the ATF-54143.

3.1 Measuring S-parameters of a FET: Thru- Reflect-Line (TRL) Calibration

Agilent specification sheets for the ATF-54143 provide S-parameters from 0.1 GHz to 18 GHz. However, displaying them on the Smith Chart shows that the S_{11} curve does a full 360^0 rotation (as shown in Figure). Using high frequency circuit simulation software such as nodal to simulate pHEMT S parameters, Bruce Veidt was able to conclude that this behavior is impossible to simulate with a FET unless something additional is connected to its input— for example, a long transmission line. Agilent's given S-parameters were clearly incorrect. Transistor S-parameters would have to be measured manually.

Accurate transistor S-parameters were obtained using TRL calibration with an Agilent 8720 ES network analyzer. The network analyzer's coaxial cables cannot be directly attached to the transistor due to the mismatch in physical size. Therefore a fixture s needed to provide an interface between the coaxial cable and the transistor, and TRL calibration is a means of eliminating fixture and cable effects on S-Parameter measurement of a non-coaxial device. TRL was chosen because it requires calibration standards that are easier to realize with microstrip (see following paragraph) at high frequencies. Short-Open-Load-Thru (SOLT) is another common way of calibrating non-oaxial devices but the required standards are not as easy to build. In particular, the load standard introduces a challenge because a purely resistive load is difficult to realize over a large bandwidth in microstrip .

Calibration fixtures were constructed using microstrip, the transmission medium of choice in building the LNA. Microstrip is a type of transmission line used in the construction of PCBs (printed circuit boards). It is commonly used in the construction of high frequency circuits due to its high bandwidth, excellent miniaturization, easy integration with chip devices, and low radiation losses. Signals are transmitted by a conducting trace on a dielectric (non conducting) substrate. The substrate rests on a conducting sheet that acts as the ground plane. Convenience lies in the simple geometry of microstrip, which allows for easy mounting of circuit components.

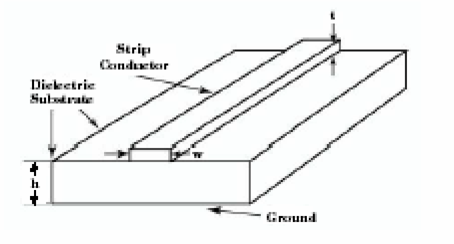


Figure 3.1 – Microsrip Geometry

The fixture is ideally lossless or“invisible.” Therefore in order to keep losses at a minimum, Rogers RO4003 microstrip was selected. RO4003 is composed of a very low loss substrate, as demonstrated in Figure 3.1. The Figure 3.2 displays insertion loss, which is obtained by measuring the gain (also known as scattering parameter S21) of the microstrip. If the microstrip were perfectly lossless, the gain would be unity (0 dB). In actuality, however, microstrip has some loss, which introduces negative gain. RO4003 is inferior to only two types of laminate on the chart: RO3003 and PTFE/woven glass. RO3003 was not chosen because it is not as rigid as RO4003, making its sturdiness questionable. PTFE is more expensive due to special high-cost fabrication processes

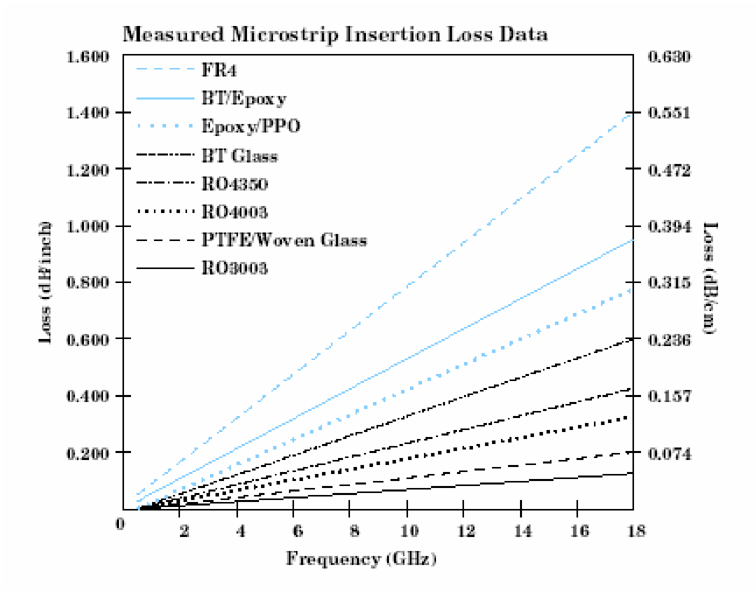


Figure 3.2 – Microstrip insertion loss.

3.2 Thru-Reflect-Line (TRL) Standards

During S-Parameter measurement, the device under test (DUT) is sandwiched between two fixture-halves. Each fixture-half terminates in a coaxial connector, acting as an interface to the coaxial cables of the network analyzer. To maximize the transparency of the fixture to the network analyzer, fixtured calibration standards are required so that the fixture itself may be calibrated out of the measurement. These standards consist of a THRU, a REFLECT and at least one LINE (depending on the range of frequencies used). The following are guidelines for building TRL standards.

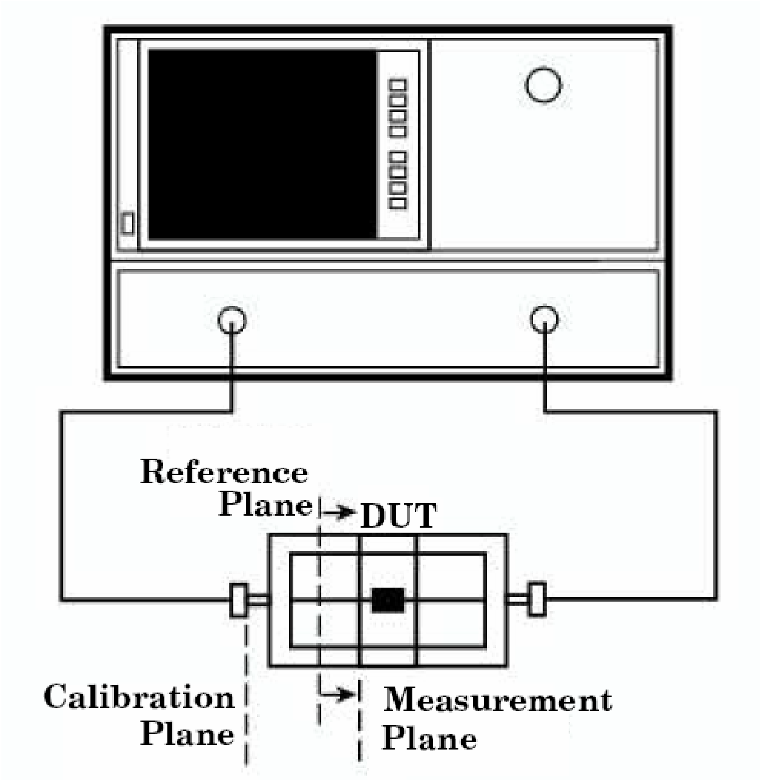


Figure 3.3 - S-parameter measurement of a non-coaxial device in fixture based

3.2.1 THRU

The THRU standard is simply the fixture by itself (i.e. the fixture-halves joined together). It consists of a 1” (2.54 cm) length of microstrip with two coaxial connectors soldered at each end. The middle of the THRU standard sets the reference plane for measurement. Recall that impedances displayed on a Smith Chart require a normalizing value. The THRU standard provides the characteristic impedance for measurements to be referenced to. In this case the standard 50 Ohms was used. The characteristic impedance of microstrip transmission line depends on not only the width of the conducting trace, but the thickness and dielectric constant of the substrate as well. The dependence of characteristic impedance on these factors can be quite complex and requires substantial computational efforts. However, there are software tools that prove to be useful. Nodal has a Zo function and Rogers Corporation offers a free program on its website to calculate characteristic impedance of microstrip lines. Using these two tools, it was calculated that RO4003, which has a dielectric constant of 3.38, should have a line width of approximately 0.080" (0.20 cm) in order to have a characteristic impedance of 50 Ohms.

3.2.2 REFLECT

The REFLECT standard can be either an open circuit or a short circuit. A perfect open or short causes 100% of an incoming voltage wave to be reflected back, which is equivalent to a 0-degree (open) or 180-degree (short) phase shift in the wave. The point of reflection establishes the reference plane (the point where each fixture-half meets the device). Because the reflection needs to occur at the reference plane, the REFLECT standard is constructed using a fixture-half that terminates in an open or a short. Due to the fringing capacitance of an open circuit created in microstrip , a short circuit was chosen for the reflect standard (fringing capacitance brings the impedance of a microstrip open circuit further away from that of an ideal open circuit). The short was constructed using vias (holes that are plated through with conductor so that they are connected to the ground plane of the microstrip).

3.2.3 LINE

Although its characteristic impedance should be equal to that of the THRU standard, each LINE standard must not have the same insertion phase as the THRU. Insertion phase is the phase of the signal at the output port relative to the input port. In other words, the LINE standard has to be a certain length longer than the THRU but not long enough that insertion phase is duplicated. The difference in insertion phase between the THRU and LINE standard must remain between ($20^0$and 160^0) $\pm n \times 180^0$ to avoid ambiguity. Optimal line length is 90^0, which is one quarter-wavelength. The usable bandwidth of a single LINE has an 8:1 ratio, meaning the upper frequency limit is 8 times that of the lowest frequency being used. Because of this bandwidth limitation, more than one line standard may be needed if the usable frequency span of a single line standard is not wide enough.

For example, the Agilent datasheet gives S-Parameter data from 500 MHz to 18 GHz. TRL calibration would require more than one LINE standard to cover this entire range. Due to the 8:1 bandwidth limitation, the upper frequency limit for LINE 1 would be 8 x 500 MHz, or 4 GHz. LINE 2 would be usable from 4 GHz to 18 GHz. The lengths of the excess transmission line (relative to the length of the THRU standard) for use within frequency range f_1 to f_2 are calculated as follows.

$$\text{length(cm)} = (15000 \cdot V\,F) / [f_1(\text{MHz}) + f_2(\text{MHz})] \text{ ------------------------------- } (3.1)$$

V F is velocity factor, or the amount the speed of electromagnetic propagation in a medium is reduced relative to that of propagation in free space. In the case of microstrip, the velocity factor depends on the effective dielectric constant of the substrate.

$$V\,F = 1 / (\sqrt{\varepsilon_{eff}}) \text{ --- } (3.2)$$

Microstrip effective dielectric constant is a function of the relative dielectric constant of the microstrip substrate and microstrip geometry. It accounts for electric fields not being fully constrained within the substrate: they “leak” into the air.

3.3 Noise parameter calculation of FET

How the equivalent circuit model of the Agilent ATF-54143 pHEMT was obtained from its measured S-parameters is documented in this chapter. Theoretically, the equivalent circuit of a FET can be determined using the FET’s measured S-parameters and simulated S-parameters over a wide frequency range. The equivalent FET circuit can then be used to calculate the FET noise parameters, which are necessary for LNA design. Simulated S-parameters can be generated from a FET equivalent circuit model using high frequency circuit simulation software. For this project, Genesys 2005 was used to simulate the S-parameters of the FET equivalent circuit. Optimization of the equivalent circuit element values was performed iteratively by least-squares fitting of simulated S-parameters to measured S-parameters of the ATF-54143 over the frequency range 0.5 - 10 GHz. To obtain initial values for the optimization process, I used a variety of sources, including values specified by Agilent for the ATF-54143 as well as already-documented values for other FETs. Finally, noise parameters were determined using Pospieszalski's method, which will be discussed in this chapter.

3.3.1 Noise Power

We know that, noise in electronic devices is generated by thermal fluctuations in electrons. The maximum power obtainable from a noisy resistor within bandwidth Δf, comes from Nyquist's derivation in 1928 of the following relation between the rms voltage of thermal noise $v_{n,rms}$ appearing across a resistor R:

$v_{n,rms} = \sqrt{(v_n^2)} = \sqrt{(4kTR\Delta f)}$ -- (3.3)

This equation can be used to model the physical resistor R as a noise-free resistor in series with a voltage source of rms voltage $\sqrt{4kTR\Delta f}$. Derived from Norton's theorem, an equivalent current source model consisting of a current source with rms current. $\sqrt{4kTG\Delta f}$ in parallel with noise-free conductance

G = 1/ R can also be used to represent a noisy resistor. The equivalent models are shown in Figure 4.1

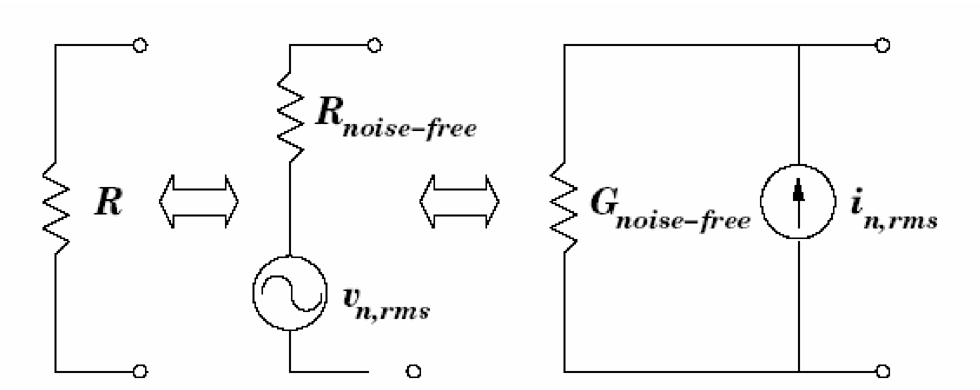


Figure 3.4 - Equivalent models of a noisy resistor R.

The maximum power obtainable from noisy resistor R. is given by the formulae of maximm power transfer theorem. With respect to $v_{n,rms}$ and R,

$P_N = kT\Delta f = v_{n,rms2} / 4R$ --- (3.4)

Maximum power transfer occurs when the load impedance is the complex conjugate of the source impedance. This can be explained by a simple network with a source generator and a load.

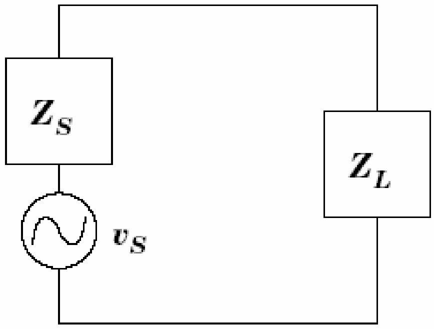


Figure 3.5 - Voltage generator driving a load.

The power dissipated in the load can be found using the following relation:

$$P = (1/2)\ |I|^2\ Re[Z_L] \text{ -- } (3.5)$$

where

$$|I| = v_s\ /\ (Z_S + Z_L) \text{ -- } (3.6)$$

Putting everything together,

$$P = (1/2)\ [v_s\ /\ (Z_S + Z_L)\ ]^2\ Re[Z_L] \text{ -- } (3.7)$$

Letting $Z = R + jX$,

$$P = (1/2)\ [\ (v_s^2\ R_L\ )\ /\ [\ (R_S + R_L\ )^2\ \ (X_S + X_L)^2\ ] \text{ ----------------------- } (3.8)$$

From the above equation, P is maximized when $Z_L = R_S - jX_S = Z^*_S$ .And when $Z_L = Z^*_S$,

$$P = v_{s,rms\ 2}\ /\ 4\ R_L \text{ --- } (3.9)$$

3.3.2 Noise Temperature and Noise Figure

A noisy two-port network can be modelled as a noiseless two-port network with additional noise power at its input. Consider a noisy two-port with noise power as - $P_{Ni} = kT_s \Delta f$ entering at its input. Input noise power P_{Ni} is amplified by the available gain of the two-port G_A, resulting in output noise power $P_{No} = G_A P_{Ni}$. If the two-port is to be modelled as noiseless, then in the model, additional noise power $kT_e \Delta f$ is added at the input, symbolizing the noisy two-port's noise power contribution referred to the input of the network. Therefore, the noisy two-port is defined as having an effective input noise temperature of T_e. The noisy two-port's input noise power contribution is amplified by the two-port's available gain G_A and shows up at the output as $(kT_e \Delta f)G_A$.

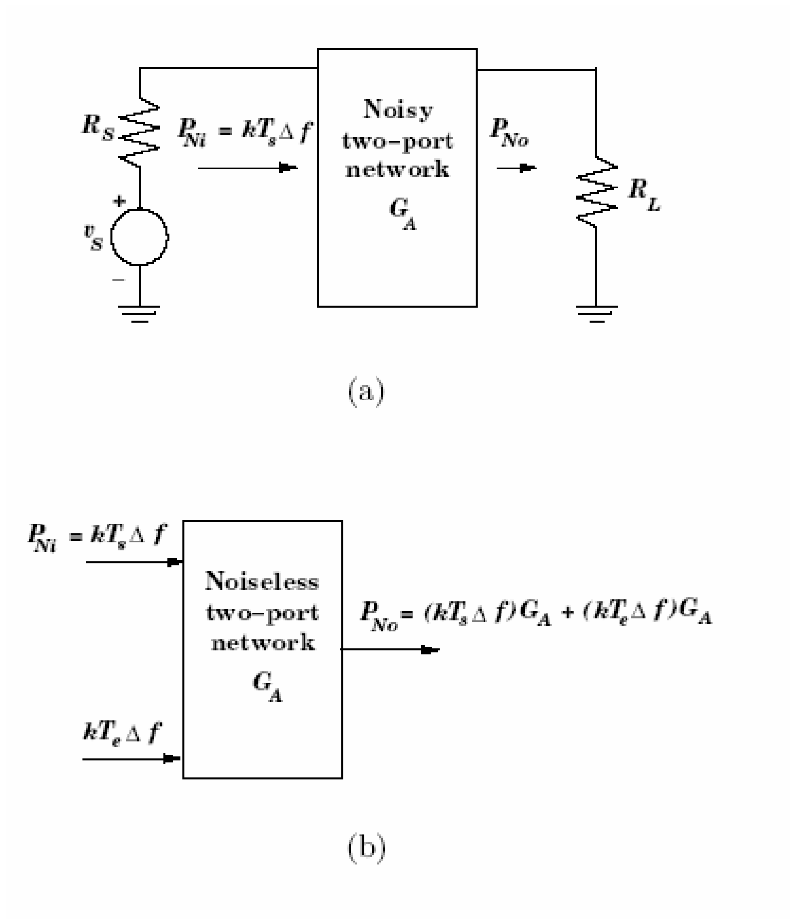


Figure 3.6 (a) Noisy two-port network. (b) Noiseless two-port network model of noisy two-port network.

Because of the additional input noise power $kT_e \Delta f$, the total output noise power P_{No} becomes a sum of two terms:

$P_{No} = G_A P_{Ni} + G_A k T_e \Delta f$ -- (3.10)

$= kT_s \Delta f G_A + kT_e \Delta f G_A$

$= kT_s(1 + T_e / T_s) \Delta f G_A$ --- (3.11)

The available gain G_A is defined as the ratio of available signal power at the output P_{So} to available signal power at the input P_{Si}:

$G_A = P_{so} / P_{Si}$ -- (3.12)

The ratio of signal-to-noise power ratio at the input to signal-to-noise ratio at the output defines the noise figure F of the two-port. In other words, noise figure is a measure of how much a signal degrades as it passes through the two-port as a result of the two-port's own noise.

The expression for the noise figure of the two-port in terms of input noise temperature T_s and additional effective input noise temperature T_e:

$(P_{si} / P_{ni}) / (P_{So}/P_{No}) = F = 1 + T_e / T_s$ ------------------------------------- (3.13)

T_e has already been defined as the noise temperature of the noisy two-port. T_s, also known as T_0, is a reference input noise temperature required to obtain the noise figure of the two-port. The standard value has been set at 290 K. The term"noise figure" has been commonly used since the 1940's, when it was defined as above by Harold Friis. In modern usage, however, the quantity F has often come to be known as "noise factor," while the same quantity in dB units, NF, is known as noise figure.

NF = 10 log F. -- (3.14)

For simplicity however, in this thesis F and NF will both be referred to as noise figure — NF being noise figure in terms of dB.

3.3.3 Noise Parameters

Noise figure is a convenient way of quantitatively describing the noise performance of an amplifier. Recall from previous Section that a major part of this project involves obtaining the noise parameters of a FET and that there are four noise parameters in total :

- T_{min} or F_{min}
- the real part of Z_{opt} or Γ_{opt}
- the imaginary part of Z_{opt} or Γ_{opt}
- equivalent noise resistance R_n or equivalent noise conductance g_n.

T_{min} and F_{min} specify the lowest noise an amplifier can have. Z_{opt} and Γ_{opt} specify what impedance the input port of the amplifier needs to see in order to achieve lowest noise. R_n and g_n are measures of the sensitivity of amplifier noise temperature T or noise figure F as Zs deviates from Z_{opt}. Noise resistance R_n and noise conductance g_n arise from the noise representation of the two-port.

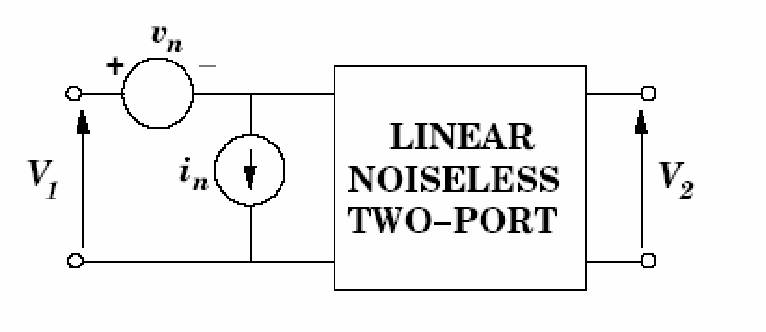


Figure 3.7 - Noise representation of noisy two-port used to define noise parameter R_n and g_n .

R_n and g_n are defined as the following :

$$R_n = |v_n|^2 / (4kT_0 \Delta f) \text{ --- } (3.15)$$

$$g_n = |i_n|^2 / (4kT_0 \Delta f) \text{ --- } (3.16)$$

where-

R_n = noise resistance (ohm)

g_n = noise conductance (siemens)

$|v_n|$ = noise voltage source amplitude (volts)

$|i_n|$ = noise current source amplitude (amperes)

k = Boltzmann's constant = 1.3807×10^{-23} J/K

T_0 = standard reference noise temperature of 290 K

Δf = bandwidth (Hz)

The noise temperature T of a two-port amplifier can be expressed as a function of the four noise parameters :

$$T = T_{min} + T_o \ (g_n / R_g) \ |Z_g - Z_{opt}|^2 \text{ --- } (3.17)$$

where

T = noise temperature of amplifier (K)

T_{min} = minimum obtainable noise temperature of amplifier(K)

T_0 = standard reference noise temperature value of 290 K

g_n = noise conductance (siemens)

R_g = real part of source impedance

Z_g = source impedance

Z_{opt} = source impedance seen by amplifier input which achieves T_{min} (K)

Since F is related to T, the noise figure F of a two-port amplifier can be expressed as a function of the four noise parameters as well.

$$F = F_{min} + \ [\ (\ 4r_n \,|\, \Gamma_S - \Gamma_{opt} \,|^2\,) \,/\, (\ (1 - |\,\Gamma_S|^2\,)\ (\ |1 + \Gamma_{opt}\,|^2))] \text{ ------------------ } (3.18)$$

$r_n = R_n / 50$ = equivalent normalized noise resistance of two-port

Γ_S = source reflection coefficient

Γ_{opt} = optimum source reflection coefficient for minimum noise

F_{min} = minimum noise figure

Constant noise figure (F) values form circles in the Γ_S-plane except for the single point F_{min}, which corresponds to $\Gamma_S = \Gamma_{opt}$

3.3.4 Using a FET Equivalent Circuit Model to Find Noise Parameters: The Pospieszalski Method

If the small-signal equivalent circuit of a FET and two particular frequency independent constants are known, then theoretically all four noise parameters (F_{min}, r_n, Re [Γ_{opt}], Im [Γ_{opt}]) can be predicted over a broad frequency range. This way of obtaining noise parameters can be attributed to Marian Pospieszalski. According to Pospieszalski, Pucel provides the most comprehensive FET noise model. However, Pucel's model requires three frequency-independent noise coefficients to be known in addition to a FET equivalent circuit model. Taking advantage of more recent work by Gupta , Pospieszalski has been able to show that accurate noise parameters may be calculated from knowledge of a FET equivalent circuit and only one frequency-independent constant . The equivalent circuit model is shown in Figure 3.8--.

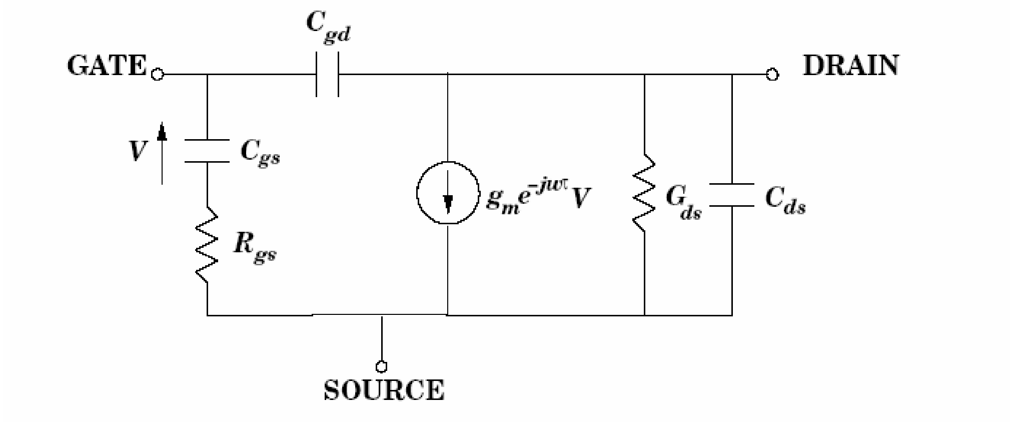


Figure 3.8 - Intrinsic noise equivalent circuit of a FET. Delay represents the transit time of electrons through the FET channel.

Initially, Pospieszalski's model requires two frequency-independent constants: the gate equivalent temperature T_g and drain equivalent temperature T_d. T_g represents the equivalent noise temperature of the gate resistance R_{gs} while T_d represents the equivalent noise temperature of the drain conductance G_{ds}. However, only one of the frequency-independent constants needs to be determined - T_d. It has been shown experimentally by Gupta that T_g can be approximated as the ambient temperature of the device. Upon obtaining an equivalent FET model and T_d, each of the four noise parameters can be expressed in terms of T_g , T_d and FET equivalent circuit parameters

$$X_{opt} = Im[Z_{opt}] = 1/ (2\pi fC_{gs}) \text{ -- } (3.19)$$

$$R_{opt} = Re [Z_{opt}] = \sqrt{ [((f_T / f)^2 (R_{gs} / G_{ds}) (T_g / T_d)) + R_{gs}^2] } \text{ ------------ } (3.20)$$

$$T_{min} = 2 (f / f_T) \sqrt{ [G_{ds}R_{gs}T_gT_d + (f/ f_T)^2 R_{gs}^2 G_{ds}^2 T_d^2 + 2 (f/ f_T)^2 R_{gs} G_{ds} T_d] }$$
$$\text{----------------------------- } (3.21)$$

$$g_n = (f/ f_T)^2 G_{ds} T_d / T_0$$
$$\text{----------------------------- } (3.22)$$

where

X_{opt} = imaginary part of optimum impedance for lowest noise ()

R_{opt} = real part of optimum impedance for lowest noise ()

T_{min} = minimum noise temperature (K)

g_n = noise conductance (siemens)

f = frequency (Hz)

f_T = cutoff frequency (Hz)

C_{gs} = gate-to-source capacitance (F)

R_{gs} = gate-to-source resistance (ohm)

R_{ds} = drain-to-source resistance (ohm)

G_{ds} = 1 / R_{ds} = drain-to-source conductance (siemens)

T_g = equivalent gate temperature (K)

T_d = equivalent drain temperature (K)

where

$$f_T = g_m / 2\pi C_{gs} \text{ --- } (3.23)$$

g_m = transconductance of FET (siemens)

The transconductance of a FET is the ratio of the change in drain current to a change in gate voltage and has units of siemens. The cutoff frequency is defined as the frequency at which FET current gain drops to unity. This FET noise model is restricted to the following condition:

$$T_{min} \leq 4NT_0$$

where

$N = R_{opt}g_n$ -- (3.24)

Values of N and T_{min} that satisfy this relation allow the model to represent a physical two-port. As will be shown later for the ATF-54143 pHEMT. As well, the Pospieszalski noise model is valid for high frequencies where 1/f or “flicker noise,” is insignificant.

CHAPTER 4

DESIGN OF LOW NOISE AMPLIFIER

In design of broad-band high frequency transistor amplifiers is a difficult and challenging problem in active network theory of great practical importance. The difficulties are largely due to the number of requirements at a time. To set one of the parameter by changing the values of the appropriate circuit elements, another parameter gets affected. Therefore in broad-band high frequency transistor amplifiers, there is need of optimization of the component values to get optimum result. At present, a rigorous design theory for broad-band transistor amplifiers is not available and designers must rely on computer-aided and experimental cut and try techniques which are often inadequate. The design of LNA mainly includes the following design steps –

(1) Design of biasing networks
(2) Design of impedance matching networks
(3) Stability calculations
(4) Design of microstrip, inductors
(5) Need of simulation for optimization-Genesys 2005 (simulation software)

4.1 Design of biasing networks (DC Biasing)

DC biasing of ATF-54143 is accomplished by the use of a voltage divider consisting of R_1 & R_2.The voltage for the divider is derived from the drain voltage which provides a form of voltage feedback through the use of R_3 to help keep drain current constant.

Resistor R_3 is calculated based on desired Vds, Ids and available power supply voltage.

$R_3 = (V_{DD} - Vds) / (Ids - I_{BB})$ -- (4.1)

Where-

V_{DD} is the power supply voltage

Vds is the device drain to source voltage.

Ids is the desired drain current.

I_{BB} is the current flowing through the R_1/ R_2 resistor voltage divider network.

From data sheet –

Here $V_{DD} = 5V$

$Vds = 3V$

$Ids = 60\ mA$

Assume I_{BB} = 10 times expected gate leakage current.

Gate leakage current = 200 μA

Therefore $I_{BB} = 10 \times 200\ \mu A = 2\ mA$

Thus,

$$R_3 = (5-3) / ((60 + 2) 10^{-3}) = 32.3\ \Omega$$

Now, R_1 is calculated as –

$R_1 = Vgs / I_{BB}$ --- (4.2)

From data sheet –

Vgs = 0.59V (Typical value)

Thus,

$$R_1 = 0.59 / 2\ mA = 295\ \Omega$$

R_2 is calculated as-

$R_2 / R_1 = (Vds – Vgs) / Vgs$ --- (4.3)

$$R_2 = ((3-0.59) 295) / 0.59 = 1205\ \Omega$$

The purpose of R_4 is to enhance the low frequency stability of the device by providing a resistive termination at low frequencies. The value of R_4 is assumed and is taken as 56 Ω.

The capacitors C_3 & C_4 provide the low frequency RF bypass for resistors R_3 & R_4. There value should be chosen carefully as C_3 & C_4 also provide a termination for low frequency mixing products. These mixing products are as a result of two or more in band signals mixing & producing third order inband distortion products. The low frequency or difference mixing products are bypassed by C_3 & C_4.For best suppression of third order distortion products, C_3 & C_4 should be 0.01 μF. in value. Smaller value of capacitor will not suppress the generation.

R_3 & R_4 provide a very important low frequency termination for the device. The resistive termination improves low frequency stability.

Thus the resistor values for biasing networks are-

$R_1 = 295\ \Omega$

$R_2 = 1205\ \Omega$

$R_3 = 32.3\ \Omega$

$R_4 = 56\ \Omega$

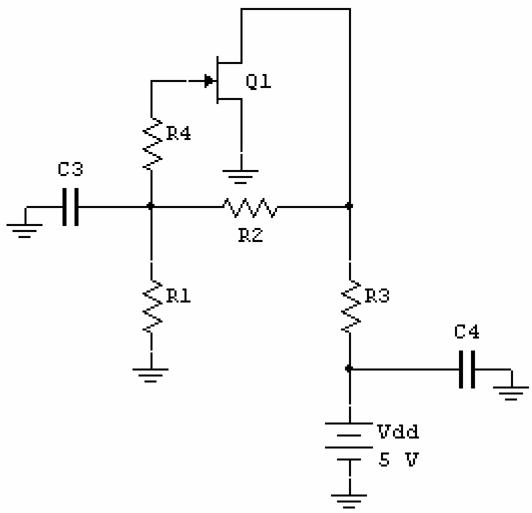


Figure 4.1 DC Biasing Network

4.2 Design of Impedance matching Networks for lowest Noise

Two of the most important noise parameters of a transistor are the real and imaginary parts of the optimum impedance Z_{opt}, which is the impedance presented to the input of the transistor which is required to obtain its lowest noise performance. When designing an LNA, the most important impedance matching consists of transforming the 50 Ω coaxial cable input to Z_{opt} by building a matching network. In an ideal world, the matching network is lossless (i.e. non-resistive) and constructed with ideal capacitors, inductors, or transmission lines. To add resistances would introduce additional noise. In the real world, inductors, capacitors and even the circuit board itself introduce small parasitic losses and add to the noise figure of the LNA. Therefore it is very important that the lowest loss components are selected.

Also, parasitics are not only composed of resistances but of small inductances and capacitances as well. While insignificant at lower frequencies, these parasitics need to be accounted for as frequency increases. It must be ensured that the LNA frequency of operation is within the frequency range that the components are specifically designed for.

The mapping of reactances, admittances and susceptances should be done on the Smith Chart. To create a matching network, a given input impedance is transformed into a desired output impedance by the use of circuit components. When added in series (parallel), inductors add positive reactance (negative susceptance), and capacitors add negative reactance (positive susceptance). Adding a length of transmission line causes movement along a circle centered on the point on the Smith Chart corresponding to the transmission line's characteristic impedance.

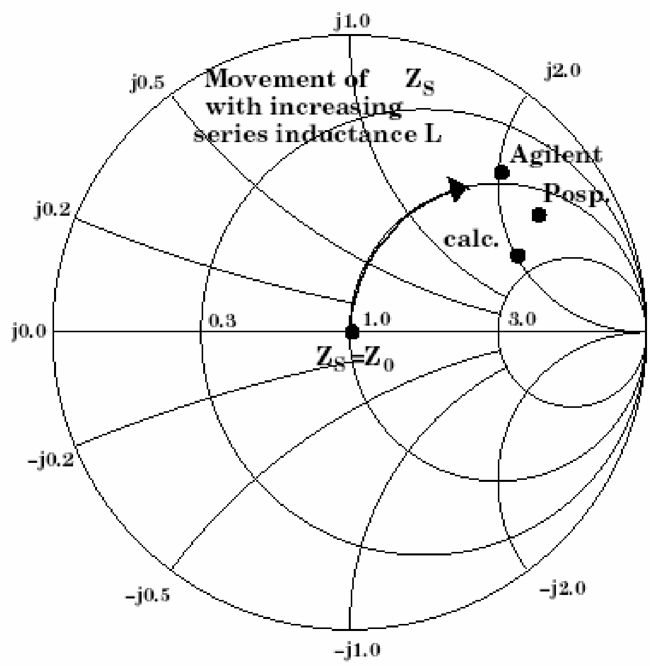


Figure 4.2 - Movement of Z_S on Smith Chart with increasing L.

A matching network can be as simple as a series inductor that translates the impedance seen by the transistor from the center of the Smith Chart (50 Ω) along a constant resistance curve, making it closer to Z_{opt}. However, one must consider the DC biasing of the transistor, where DC voltages are applied to the gate and drain to keep the transistor in the desired state of operation. For instance, to direct the DC voltage specifically to the gate, a blocking capacitor at the input of the LNA is needed to keep the DC signal from leaking into the generator. A blocking capacitor acts as an open circuit at low frequencies and as a short circuit at high frequencies. To keep the RF signal from leaking into the biasing circuit, an RF choke is used. An RF choke is an inductance that acts as an open circuit at high frequencies and as a short circuit at low frequencies. To provide further isolation between the LNA circuit and the biasing circuit, RF bypass capacitors were added. An RF bypass capacitor acts like a short circuit at high frequencies. As well, a 50 Ω resistor was added after the RF choke to improve stability at lower frequencies. The stabilizing resistor forces Z_S to be closer to the center of the Smith Chart at lower frequencies.

The initial configuration of the input matching network design appears su_cient in getting Z_S closer to Z_{opt}. However, the input matching network needs to be modelled more thoroughly during the design process. Not only do the values of inductors, capacitances and resistances need to be considered, but so do the widths and lengths of conducting traces of microstrip between the components. As well, the number of tuning elements can greatly complicate the building and testing of a working LNA. When dealing with fine-tuning the LNA in order to achieve lowest noise, it helps to rely on fewer tuning elements in the input matching network. A better matching network would be one composed of a large DC block and large RF choke. This way, there is good isolation between the LNA circuit and the biasing circuit and Z_S remains close to the center of the Smith Chart until tuning elements are added. As well, restricting the number of bypass capacitors to just a single large capacitance placed before the biasing circuit greatly improves stability. Following figure summarizes a more practical and straightforward input matching network.

To meet the 33 dB minimum gain requirement of a GMRT Radio Telescope frontend receiver, the LNA design consists of two stages of amplification using two ATF-54143 transistors. In the initial design, three matching networks were included to optimize the first stage for low noise and the second stage for a compromise between low noise and high gain. Located between the LNA input and the input of the first transistor, the input matching network transforms 50Ω to Z_{opt}.

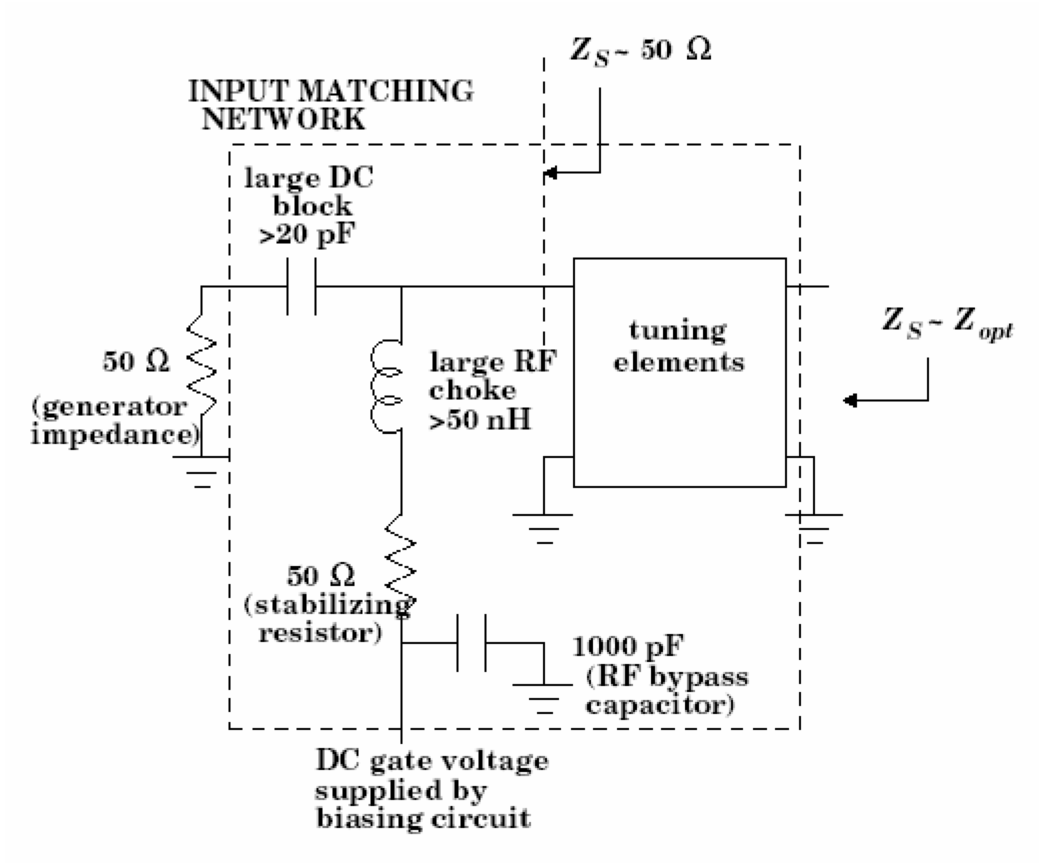


Figure 4.3 - A better input matching network.

Located between the output of the first transistor and the input of the second transistor, the interstage matching network presents the output impedance of the first transistor. The interstage matching network transforms Z_{out1} to an impedance Z_{opt} which is presented to the input of the second transistor for lower noise.

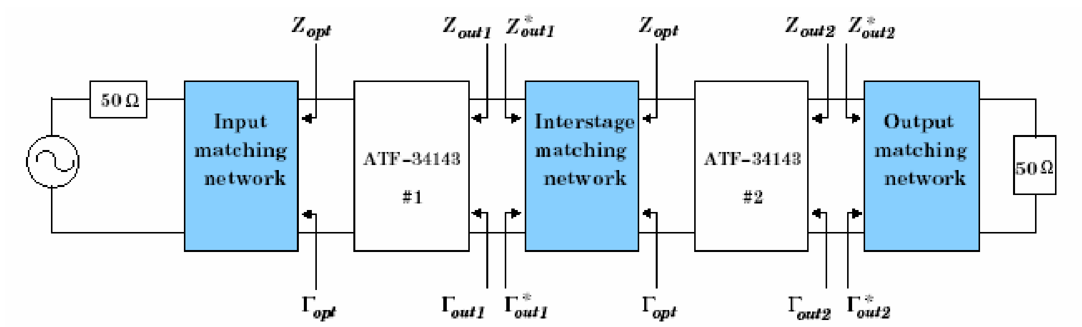


Figure 4.4 - Matching networks for a two-stage LNA.

Finally, the output matching network transforms the output impedance of the second transistor Z_{out2} closer to the characteristic transmission line impedance 50 Ω in order to minimize reflections when the output of the LNA is connected to a coaxial cable. A shunt resistance of 330 Ω with small capacitance of 10 pF was added to the drain of the two transistors to improve stability at higher frequencies. Small lengths of inductive transmission line were added to the source leads to improve input return loss and stability. The ATF-54143 is packaged so that there are two source leads. Specifically, approximately 1nH of extra inductance was added to source lead.. The total source inductance of each transistor is approximately 1 nH. Transmission line inductances are used for the source inductance.

Genesys 2005 was used to simulate the performance of the LNA and the resulting simulated noise figure was 0.3 dB (20 K). The 33 dB minimum gain requirement was satisfied, input and output return losses were on the order of 10 dB and the stability K factor was greater than 1 up to frequencies on the order of 10 GHz. However, when the LNA design was physically realized, the noise measurements were unsatisfactory, with noise figures on the order of 1 dB. There were also problems with oscillations. After much trial and error via replacement and shifting of components especially at the input of the first stage, a noise figure of 0.4 dB (30 K) was finally achieved.

4.2.1 Design rules for Impedance matching Networks

If certain rules are followed we can design impedance matching networks speedily. These rules are merely a guideline to be followed in the matching circuit design process. They will never replace the theoretical and p[ractical understandings that go into taking them. These rules are summarized as follows.

(1) Use a ZY smith chart.
(2) Always start off from the source end and travels towards load.
(3) Always move on a constant Z or constant G circle in such a way as to arrive eventually at the center of the smith chart
(4) Each motion along a constant Z or constant G circle gives the value of a reactive element.

(5) Moving on a constant R-circle yields series reactive elements, whereas moving on a constant G circle yields shunt reactive elements.
(6) The direction of travel (or motion) on a constant Z or constant G circle determines the type of element to be used : a capacitor or an inductor.

Rules 5 and 6 lead to the following additional two rules.

(7) When the motion is upward, it usually corresponds to a series or a shunt inductor.
(8) When the motion is downward, it usually corresponds to a series or a shunt capacitor.

4.2.2 Design of Impedance matching networks

We have to design a broadband low noise amplifier within the frequency range 500 MHz – 900 MHz. For that, we have to design impedance matching networks which will allow that band to pass and fulfill the requirements of the low noise amplifier parameters.

The impedance matching networks are designed for the three frequencies (500 MHz, 700 MHz, 900 MHz) for which S-parameters of the ATF 54143 are available from the datasheet.

We have selected V_{DS} = 3 V and I_{DS} = 60 mA for ATF 54143 due to low noise figure for these voltages.

(I) Impedance matching network for 500 MHz frequency :

From the data sheet of ATF 54143, for 500 MHz frequency and for $V_{DS} = 3$ V and $I_{DS} = 60$ mA, we have

$\Gamma_{opt} = 0.34 \angle 42.3^0 = 0.25 + 0.23\,j$

$S_{22} = 0.4 \angle -58.8^0 = 0.2 - 0.\,34\,j$

$Z_o = 50\ \Omega$

(1) Design of input impedance matching network $Z_o \Rightarrow Z_{opt}$

We have, Z_{opt} given by,

$$Z_{opt} = Z_o\,[\,(\,1 + \Gamma_{opt}\,) / (1 - \Gamma_{opt}\,)\,]$$

$$= 50\,[\,(\,1 + 0.25 + 0.23\,j\,) / (1 - (\,0.25 + 0.23\,j\,)\,)\,]$$

$$= 50\,[\,(\,1.25 + 0.23\,j\,) / (0.75 - 0.23\,j\,)\,]$$

$$= 50\,[\,(\,1.27 \angle 10.42^0\,) / (0.78 \angle -17.05^0\,)\,]$$

$$= 50\,(\,1.63 \angle 27.47^0\,) = 81.5 \angle 27.47^0$$

$$= (\,72.31\ + 37.6\,j\,)$$

Thus input impedance matching network is,

$Z_o \Rightarrow Z_{opt}$

$50\ \Omega \Rightarrow\ 72.31\ + 37.6\,j$ (or $81.5 \angle 27.47^0$)

Now normalized impedances are,

$\mathcal{Z}_o = Z_o / Z_o = 50 / 50 = (1 + o\,j\,)\ \Omega$

$y_o = 1 / (\,1 + 0\,j\,) = 1 \angle 0^0 = 1 + 0\,j$

$\mathcal{Z}_{opt} = Z_{opt} /\ Z_o = (\,72.31 + 37.6\,j\,) / 50$

$$\mathcal{Z}_{opt}\ = (\,81.5 \angle 27.47^0\,) / 50$$

$$= 1.63 \angle 27.47^0 = 1.45 + 0.75\,j \Rightarrow 1.5 + 0.8\,j$$

$$y_{opt} = 1 / \mathcal{Z}_{opt} = 1 / (\,1.63 \angle 27.47^0\,)$$

$$= 0.61 \angle -27.47^0 = 0.54 - 0.28\,j \Rightarrow 0.5 - 0.3\,j$$

Now, we will locate the points $\mathcal{Z}_{opt}$ and $\mathcal{Z}_o$ on the smith chart and find out the path

$\mathcal{Z}_o \Rightarrow \mathcal{Z}_{opt}$

The required path is as shown in figure 4.5. The path is so selected that the impedance matching network will also be high pass filter. The path is –

$\mathcal{Z}_o\ \rightarrow \mathcal{Z}_A \rightarrow \mathcal{Z}_{opt}$

The point ‘A’ is marked on the smith chart as shown in figure. The normalized impedance of point A is –

$z_A = (1 – 1 j) = 1.4 \angle - 45^0$

$y_A = 1 / (1.4 \angle - 45^0) = 0.7 \angle\ 45^0 = (0.5 + 0.5 j)$

(1) For the first path, $z_o \rightarrow z_A$; there is series capacitance

$\therefore 1 / (j\omega Z_oC) = z_A – z_o$

$= (1 – 1 j) - (1 + o j) = 1 j$

$\therefore\ C = 1 / (2\pi \text{ x } 500\ 10^6 \text{ x } 50 \text{ x } 1) = 6.36$ PF.

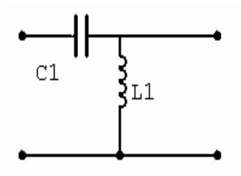
(2) For the second path, $z_A \rightarrow z_{opt}$; there is shunt inductance.

$\therefore Z_o / (j\omega L) = y_{opt} – y_A$

$= (0.5 – 0.3 j) – (0.5 + 0.5 j) = -0.8 j$

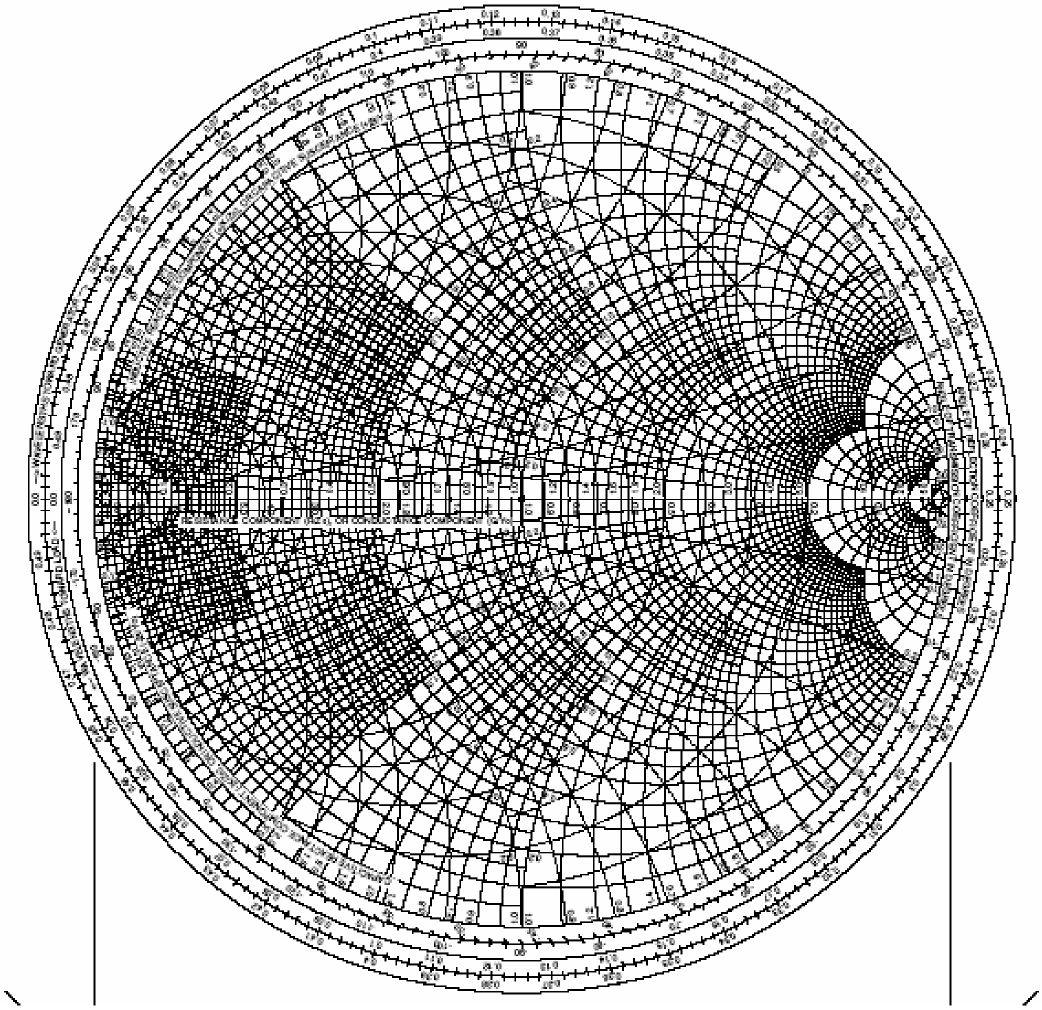
$\therefore L = Z_o / (\omega \text{ x } 0.8) = 50 / (2\pi \text{ x } 500\ 10^6 \text{ x } 0.8) = 19.89$ nH

Thus the required input impedance matching network will be –



Where C1 = 6.36 PF

L1 = 19.89 nH



(2) Design of intermediate impedance matching network $Z_{out} \Rightarrow Z_{opt}$

For 500 MHz frequency, S_{22} is -

$S_{22} = (0.4 \angle -58.8^0) = (0.2 - 0.34 j)$

We have, Z_{out} given by the formula,

$$Z_{out} = Zo [(1 + S_{22}) / (1 - S_{22})]$$

$$= 50 [(1 + (0.2 - 0.34 j)) / (1 – (0.2 - 0.34 j))]$$

$$= 50 [(1.2 - 0.34 j)) / (0.8 + 0.34 j))]$$

$$= 50 [(1.24 \angle -15.82^0) / (0.87 \angle 23.03^0)]$$

$$= 50 (1.43 \angle -38.85^0)$$

$$= 71.5 \angle -38.85^0 = (55.68 – 44.85 j)$$

Now, design of output impedance matching network, $Z_{out} \Rightarrow Z_{opt}$

i.e. $(55.68 – 44.85 j) \Rightarrow (72.31 + 37.6 j)$

The normalized impedances are –

We have,

$\mathcal{Z}_{opt} = 1.63 \angle 27.47^0 = 1.45 + 0.75 j \Rightarrow 1.5 + 0.8 j$

$y_{opt} = 0.61 \angle - 27.47^0 = 0.54 - 0.28 j \Rightarrow 0.5 – 0.28 j$

$$\mathcal{Z}_{out} = Z_{out} / Zo = (71.5 \angle -38.85^0) / 50$$

$$= 1.43 \angle -38.85^0$$

$$= 1.1 - 0.9 j \Rightarrow 1.1 - 1 j$$

$$y_{out} = 1 / (1.43 \angle -38.85^0)$$

$$= 0.7 \angle 38.85^0 = 0.55 + 0.44 j \Rightarrow 0.5 + 0.4 j$$

The points ‘A’ and ‘B’ are marked on the smith chart as shown in figure 4.6. The path is –

$\mathcal{Z}_{out} \rightarrow \mathcal{Z}_A \rightarrow \mathcal{Z}_B \rightarrow \mathcal{Z}_{opt}$

The normalized impedances of point A and B are as –

$\mathcal{Z}_A = (1 + 1 j) = 1.4 \angle 45^0$

$y_A = 1 / (1.4 \angle 45^0) = 0.7 \angle - 45^0 = (0.5 - 0.5 j)$

$\mathcal{Z}_B = (1 – 0.8 j) = 1.3 \angle -38.66^0$

$y_B = 1 / (1.3 \angle -38.66^0) = 0.76 \angle 38.66^0 = 0.5 + 0.5 j$

(1) For the first path, $\mathcal{Z}_{out} \rightarrow \mathcal{Z}_A$; there is shunt inductance

$\therefore \; Z_o / (j\omega L) = y_A – y_{out}$

$= (0.5 - 0.5 j) - (0.5 + 0.4 j) = - 0.9 j$

$\therefore L = Z_o / (\omega \times 0.9) = 50 / (2\pi \times 500\ 10^6 \times 0.9) = 17.68$ nH

(2) For the second path, $z_A \rightarrow z_B$; there is series capacitance.

$\therefore 1 / (j\omega Z_o C) = z_B - z_A$

$= (1 – 0.8 j) – (1 + 1 j) = -1.8 j$

$\therefore C = 1 / (\omega Z_o \times 1.8) = 1 / (2\pi \times 500\ 10^6 \times 50 \times 1.8) = 3.54$ PF.

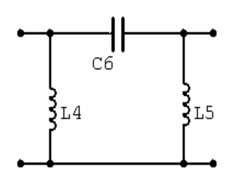
(3) For the third path, $z_B \rightarrow z_{opt}$; there is shunt inductance

$\therefore Z_o / (j\omega L) = y_{opt} - y_B$

$= (0.5 – 0.28 j) - (0.5 + 0.5 j) = - 0.78 j$

$\therefore L = Z_o / (\omega \times 0.78) = 50 / (2\pi \times 500\ 10^6 \times 0.78) = 20.4$ nH

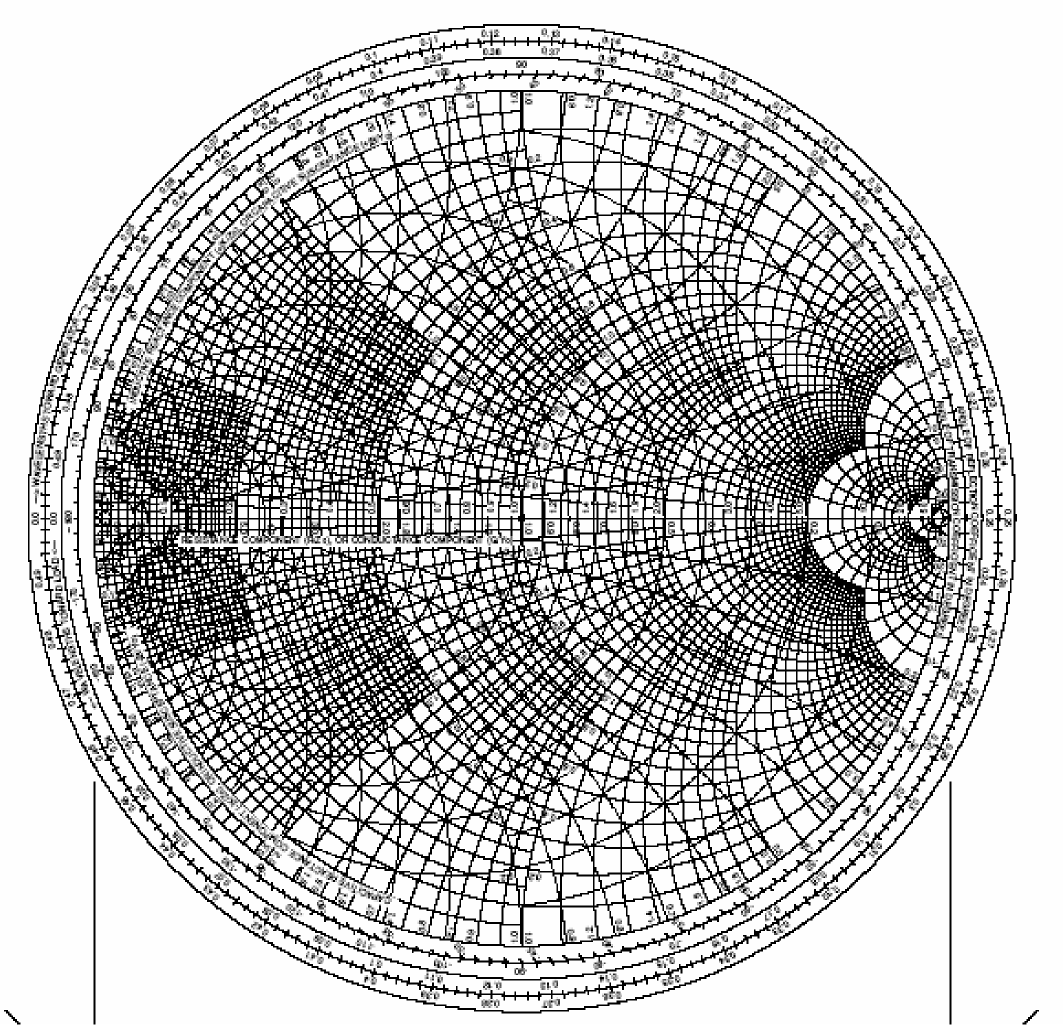
Thus the required intermediate impedance matching network will be –



Where L4 = 17.68 nH

L5 = 20.4 nH

C6 = 3.54 PF



(3) Design of output impedance matching network $Z_{out} \Rightarrow Z_o$

We have,

$Z_{out} = 1.43 \angle -38.85^0 = 1.1 - 0.9\,j \Rightarrow (1 - 1\,j)$

$y_{out} = 0.7 \angle 38.85^0 = 0.55 + 0.44\,j \Rightarrow (0.5 + 0.44\,j)$

$Z_o = Z_o / Z_o = 50 / 50 = (1 + o\,j)$

$y_o = 1 / (1 + 0\,j) = 1 \angle 0^0 = (1 + 0\,j)$

The point 'A' is marked on the smith chart as shown in figure 4.7. The path is –

$Z_{out} \rightarrow Z_A \rightarrow Z_o$

The normalized impedance of point A is –

$Z_A = (1 + 1\,j) = 1.4 \angle 45^0$

$y_A = 1 / (1.4 \angle 45^0) = 0.7 \angle - 45^0 = (0.5 - 0.5\,j)$

(1) For the first path, $Z_{out} \rightarrow Z_A$; there is shunt inductance

$\therefore\ Z_o / (j\omega L) = y_A - y_{out}$

$= (0.5 - 0.5\,j) - (0.5 + 0.44\,j) = - 0.94\,j \Rightarrow -1j$

$\therefore L = Z_o / (\omega \times 1) = 50 / (2\pi \times 500\ 10^6 \times 1) = 15.91$ nH

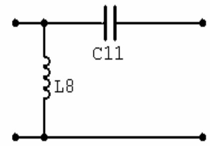
(2) For the second path, $Z_A \rightarrow Z_o$; there is series capacitance.

$\therefore 1 / (j\omega Z_oC) = Z_o - Z_A$

$= (1 + 0\,j) – (1 + 1\,j) = -1\,j$

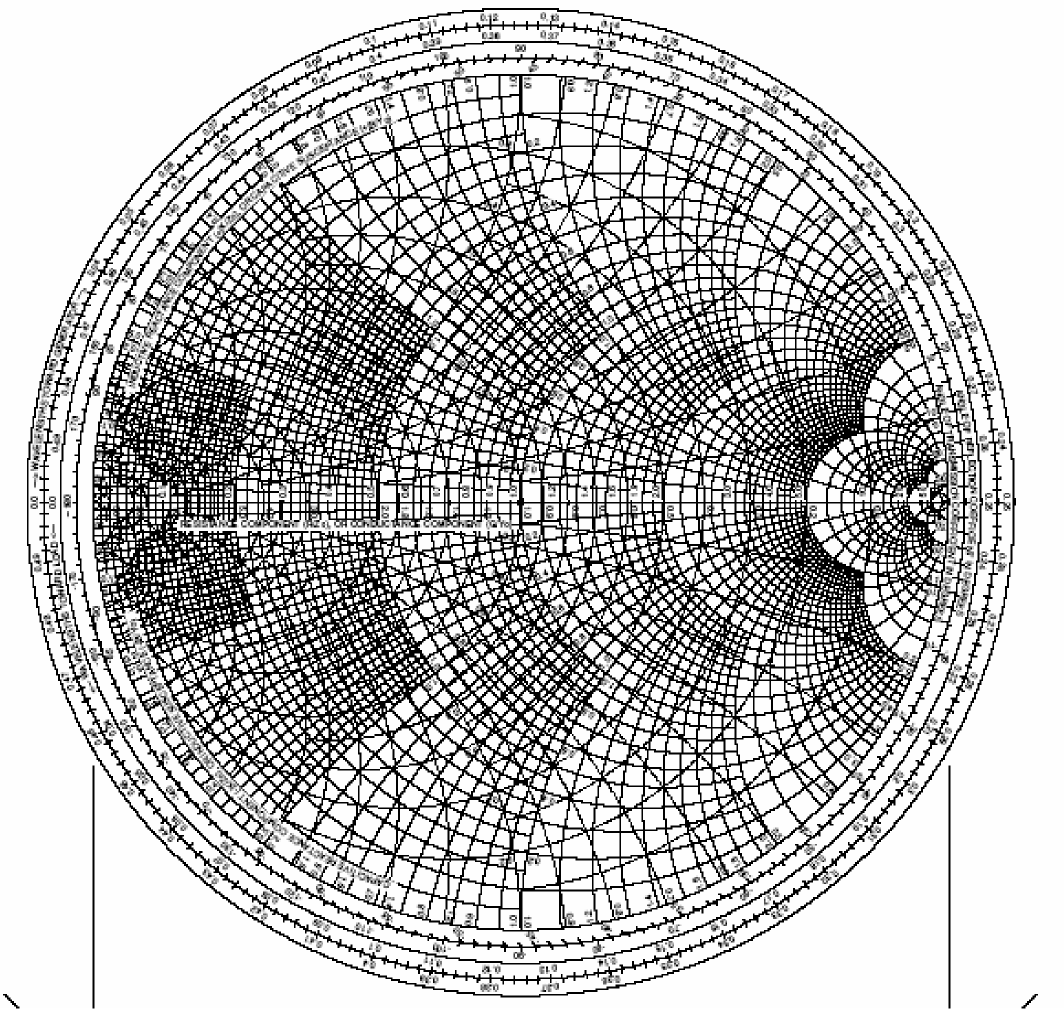
$\therefore\ C = 1 / (\omega Z_o \times 1) = 1 / (2\pi \times 500\ 10^6 \times 50 \times 1) = 6.36$ PF.

Thus the required output impedance matching network will be –



Where L8 = 15.9 nH

C11 = 6.36 PF



(II) Impedance matching network for 700 MHz frequency :

From the data sheet of ATF 54143, for 700 MHz frequency and for $V_{DS} = 3$ V and $I_{DS} = 60$ mA, we have

$\Gamma_{opt} = 0.33 \angle 52.6^0 = 0.2 + 0.26\,j$

$S_{22} = 0.35 \angle --71.3^0 = 0.11 - 0.\,33\,j$

$Z_o = 50\ \Omega$

(1) Design of input impedance matching network $Z_o \Rightarrow Z_{opt}$

We have, Z_{opt} given by,

$$Z_{opt} = Z_o\,[\,(1 + \Gamma_{opt}) / (1 - \Gamma_{opt})\,]$$
$$= 50\,[\,(1 + 0.2 + 0.26\,j) / (1 – (0.2 + 0.26\,j))\,]$$
$$= 50\,[\,(1.2 + 0.26\,j) / (0.8 - 0.26\,j)\,]$$
$$= 50\,[\,(1.23 \angle 12.22^0) / (0.84 \angle -18^0)\,]$$
$$= 50\,(1.46 \angle 30.22^0) = 73 \angle 30.22^0$$
$$= (63 + 36.74\,j)$$

Thus input impedance matching network is,

$Z_o \Rightarrow Z_{opt}$

$50\ \Omega \Rightarrow (63 + 36.74\,j)$ or $(73 \angle 30.22^0)$

Now normalized impedances are,

$\mathcal{Z}_o = Z_o / Z_o = 50 / 50 = (1 + o\,j)\ \Omega$

$y_o = 1 / (1 + 0\,j) = 1 \angle 0^0 = 1 + 0\,j$

$\mathcal{Z}_{opt} = Z_{opt} / Z_o = (73 \angle 30.22^0) / 50$

$$\mathcal{Z}_{opt} = (73 \angle 30.22^0) / 50$$
$$= 1.46 \angle 30.22^0 = 1.26 + 0.74\,j$$
$$y_{opt} = 1 / \mathcal{Z}_{opt} = 1 / (1.46 \angle 30.22^0)$$
$$= 0.68 \angle - 30.22^0 = 0.59 - 0.34\,j \Rightarrow 0.6 – 0.3\,j$$

Now, we will locate the points $\mathcal{Z}_{opt}$ and $\mathcal{Z}_o$ on the smith chart and find out the path $\mathcal{Z}_o \Rightarrow \mathcal{Z}_{opt}$

The required path is as shown in figure 4.8. The path is so selected that the impedance matching network will also be high pass filter. The path is –

$\mathcal{Z}_o \rightarrow \mathcal{Z}_A \rightarrow \mathcal{Z}_{opt}$

The point 'A' is marked on the smith chart as shown in figure. The normalized impedance of point A is –

$z_A = (1 – 0.8 j) = 1.28 \angle - 38.65^0$

$y_A = 1 / (1.28 \angle - 38.65^0) = 0.8 \angle\ 38.65^0 = (0.6 + 0.5 j)$

(1) For the first path, $z_o \rightarrow z_A$; there is series capacitance

$\therefore 1 / (j\omega Z_oC) = z_A – z_o$

$= (1 – 0.8 j) - (1 + o j) = - 0.8 j$

$\therefore\ C = 1 / (2\pi \times 700\ 10^6 \times 50 \times 0.8) = 5.68$ PF.

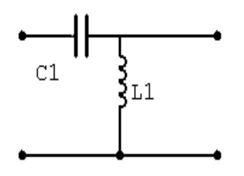
(2) For the second path, $z_A \rightarrow z_{opt}$; there is shunt inductance.

$\therefore Z_o / (j\omega L) = y_{opt} – y_A$

$= (0.6 – 0.3 j) – (0.6 + 0.5 j) = - 0.8 j$

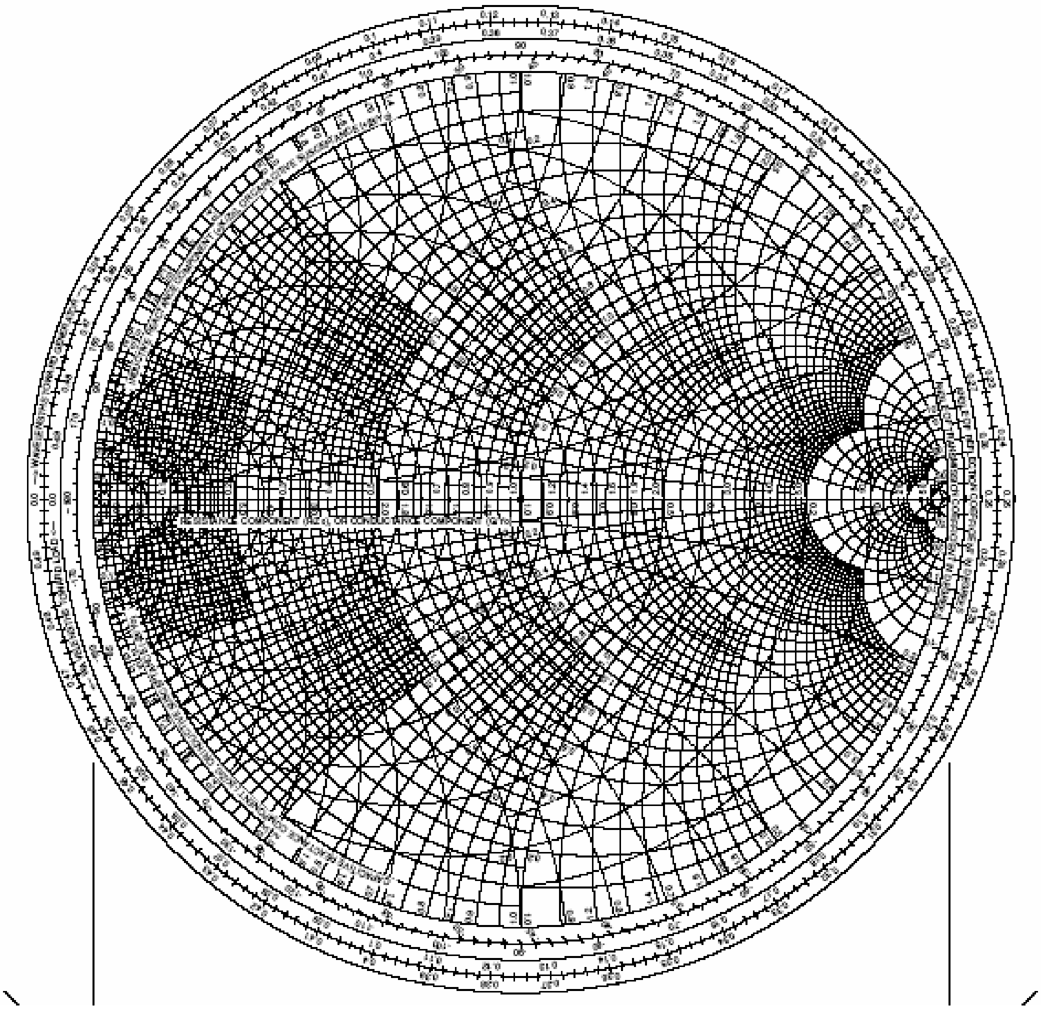
$\therefore L = Z_o / (\omega \times 0.8) = 50 / (2\pi \times 700\ 10^6 \times 0.8) = 14.21$ nH

Thus the required input impedance matching network will be –



Where C1 = 5.68 PF

L1 = 14.21 nH



(2) Design of intermediate impedance matching network $Z_{out} \Rightarrow Z_{opt}$

For 700 MHz frequency, S_{22} is -

$S_{22} = (0.35 \angle -71.3^0) = (0.11 - 0.33 j)$

We have, Z_{out} given by the formula,

$$Z_{out} = Z_o [(1 + S_{22}) / (1 - S_{22})]$$
$$= 50 [(1 + (0.11 - 0.33 j)) / (1 – (0.11 - 0.33 j))]$$
$$= 50 [(1.11 - 0.33 j)) / (0.89 + 0.33 j))]$$
$$= 50 [(1.16 \angle -16.56^0) / (0.94 \angle 20.34^0)]$$
$$= 50 (1.234 \angle -36.9^0)$$
$$= 61.7 \angle -36.9^0 = (49.34 – 37 j)$$

Now, design of output impedance matching network, $Z_{out} \Rightarrow Z_{opt}$

i.e. $(49.34 – 37 j) \Rightarrow (63 + 36.74 j)$

The normalized impedances are –

We have,

$z_{opt} = 1.46 \angle 30.22^0 = 1.26 + 0.74 j \Rightarrow 1.3 + 0.7 j$

$y_{opt} = 0.68 \angle - 30.22^0 = 0.59 - 0.34 j \Rightarrow 0.6 – 0.3 j$

$$z_{out} = Z_{out} / Z_o = (61.7 \angle -36.9^0) / 50$$
$$= 1.234 \angle - 36.9^0$$
$$= 0.98 - 0.74 j \Rightarrow 1 – 0.7 j$$
$$y_{out} = 1 / (1.234 \angle - 36.9^0)$$
$$= 0.81 \angle 36.9^0 = 0.65 + 0.48 j \Rightarrow 0.7 + 0.5 j$$

The points ‘A’ and ‘B’ are marked on the smith chart as shown in figure 4.9. The path is – $z_{out} \rightarrow z_A \rightarrow z_B \rightarrow z_{opt}$

The normalized impedances of point A and B are as –

$z_A = (0.6 + 0.7 j) = 0.92 \angle 49.4^0$

$y_A = 1 / (0.92 \angle 49.4^0) = 1.08 \angle - 49.4^0 = (0.7 - 0.8 j)$

$z_B = (0.6 – 0.8 j) = 1 \angle - 53.13^0$

$y_B = 1 / (1 \angle - 53.13^0) = 1 \angle 53.13^0 = 0.6 + 0.8 j$

(1) For the first path, $z_{out} \rightarrow z_A$; there is shunt inductance

$$\therefore \; Z_o / (j\omega L) = y_A – y_{out}$$
$$= (0.7 - 0.8 j) - (0.7 + 0.5 j) = - 1.3 j$$

$\therefore L = Z_o / (\omega \times 1.3) = 50 / (2\pi \times 700\ 10^6 \times 1.3) = 8.74$ nH

(2) For the second path, $z_A \rightarrow z_B$; there is series capacitance.

$\therefore 1 / (j\omega Z_oC) = z_B - z_A$

$= (0.6 – 0.8 j) – (0.6 + 0.7 j) = -1.5 j$

$\therefore$ $C = 1 / (\omega Z_o \times 1.5) = 1 / (2\pi \times 700\ 10^6 \times 50 \times 1.5) = 3.03$ PF.

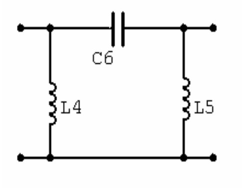
(3) For the third path, $z_B \rightarrow z_{opt}$; there is shunt inductance

$\therefore$ $Z_o / (j\omega L) = y_{opt} - y_B$

$= (0.6 – 0.3 j) - (0.6 + 0.8 j) = - 1.1 j$

$\therefore L = Z_o / (\omega \times 1.1) = 50 / (2\pi \times 700\ 10^6 \times 1.1) = 10.33$ nH

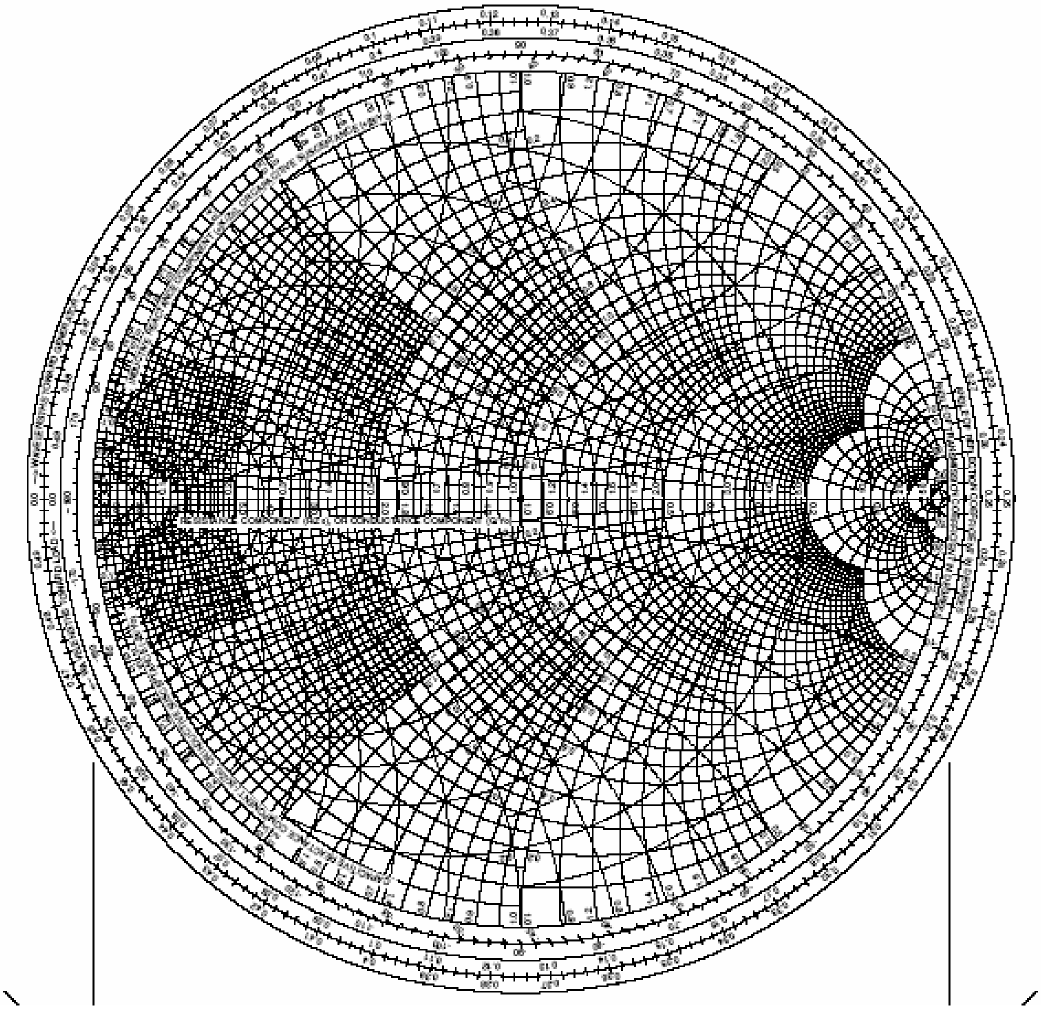
Thus the required intermediate impedance matching network will be –



Where L4 = 8.74 nH

L5 = 10.33 nH

C6 = 3 PF



(3) Design of output impedance matching network $Z_{out} \Rightarrow Z_o$

We have,

$\mathcal{Z}_{out} = 1.234 \angle -36.9^0 = 1 - 0.7\,j$

$y_{out} = 0.81 \angle 36.9^0 = 0.65 + 0.48\,j \Rightarrow 0.7 + 0.5\,j$

$\mathcal{Z}_o = Z_o / Z_o = 50 / 50 = (1 + o\,j)$

$y_o = 1 / (1 + 0\,j) = 1 \angle 0^0 = (1 + 0\,j)$

The point 'A' is marked on the smith chart as shown in figure 4.10. The path is –

$\mathcal{Z}_{out} \rightarrow \mathcal{Z}_A \rightarrow \mathcal{Z}_o$

The normalized impedance of point A is –

$\mathcal{Z}_A = (1 + 0.65\,j) = 1.2 \angle 33.02^0$

$y_A = 1 / (1.2 \angle 33.02^0) = 0.83 \angle -33.02^0 = (0.7 - 0.45\,j)$

(1) For the first path, $\mathcal{Z}_{out} \rightarrow \mathcal{Z}_A$; there is shunt inductance

$\therefore\ Z_o / (j\omega L) = y_A - y_{out}$

$= (0.7 - 0.45\,j) - (0.7 + 0.5\,j) = -0.95\,j$

$\therefore L = Z_o / (\omega \times 0.95) = 50 / (2\pi \times 700\ 10^6 \times 0.95) = 11.96\ nH \Rightarrow 12\ nH$

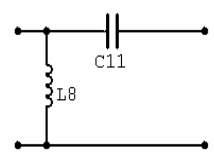
(2) For the second path, $\mathcal{Z}_A \rightarrow \mathcal{Z}_o$; there is series capacitance.

$\therefore 1 / (j\omega Z_o C) = \mathcal{Z}_o - \mathcal{Z}_A$

$= (1 + 0\,j) - (1 + 0.65\,j) = -0.65\,j$

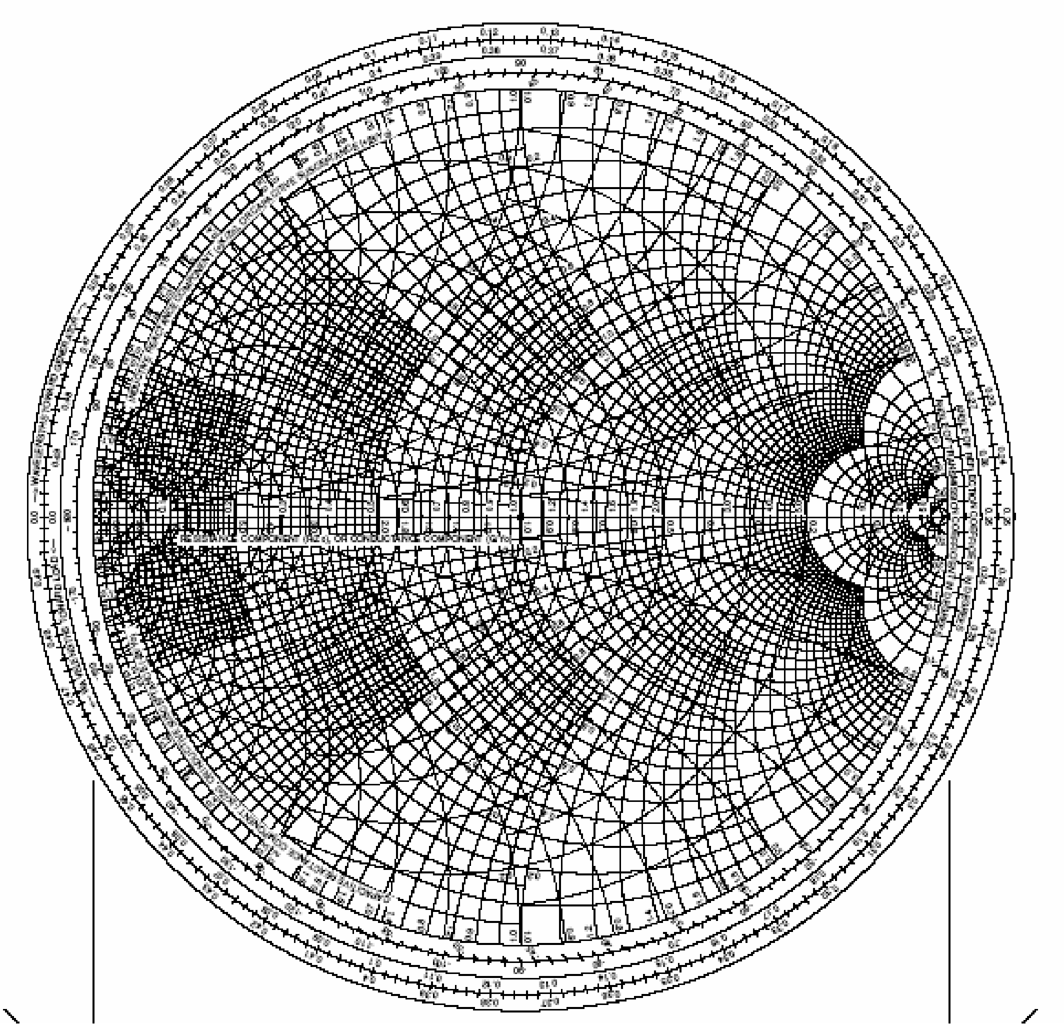
$\therefore\ C = 1 / (\omega Z_o \times 0.65) = 1 / (2\pi \times 700\ 10^6 \times 50 \times 0.65) = 6.99\ PF \Rightarrow 7\ PF$

Thus the required output impedance matching network will be –



Where L8 = 12 nH

C11 = 7 PF



(III) Impedance matching network for 900 MHz frequency :

From the data sheet of ATF 54143, for 900 MHz frequency and for $V_{DS} = 3$ V and $I_{DS} = 60$ mA, we have

$\Gamma_{opt} = 0.32 \angle 62.8^0 = 0.14 + 0.28 j$

$S_{22} = 0.29 \angle --83.8^0 = 0.03 - 0.29 j \Rightarrow 0.03 - 0.3 j$

$Z_o = 50\ \Omega$

(1) Design of input impedance matching network $Z_o \Rightarrow Z_{opt}$

We have, Z_{opt} given by,

$Z_{opt} = Z_o [(1 + \Gamma_{opt}) / (1 - \Gamma_{opt})]$

$= 50 [(1 + 0.14 + 0.28 j) / (1 – (0.14 + 0.28 j))]$

$= 50 [(1.14 + 0.28 j) / (0.86 - 0.28 j)]$

$= 50 [(1.17 \angle 13.8^0) / (0.9 \angle -18.03^0)]$

$= 65 \angle 31.83^0$

$= (55.26 + 34.28 j)$

Thus input impedance matching network is,

$Z_o \Rightarrow Z_{opt}$

$50\ \Omega \Rightarrow (55.26 + 34.28 j)$ or $(65 \angle 31.83^0)$

Now normalized impedances are,

$\mathcal{Z}_o = Z_o / Z_o = 50 / 50 = (1 + o j)\ \Omega$

$y_o = 1 / (1 + 0 j) = 1 \angle 0^0 = 1 + 0 j$

$\mathcal{Z}_{opt} = Z_{opt} / Z_o = (65 \angle 31.83^0) / 50$

$\mathcal{Z}_{opt} = (65 \angle 31.83^0) / 50$

$= 1.3 \angle 31.83^0 = 1.1 + 0.6 j$

$y_{opt} = 1 / \mathcal{Z}_{opt} = 1 / (1.3 \angle 31.83^0)$

$= 0.77 \angle - 31.81^0 = 0.65 - 0.4 j \Rightarrow 0.7 - 0.4 j$

Now, we will locate the points $\mathcal{Z}_{opt}$ and $\mathcal{Z}_o$ on the smith chart and find out the path $\mathcal{Z}_o \Rightarrow \mathcal{Z}_{opt}$

The required path is as shown in figure 4.11. The path is so selected that the impedance matching network will also be high pass filter. The path is –

$\mathcal{Z}_o \rightarrow \mathcal{Z}_A \rightarrow \mathcal{Z}_{opt}$

The point 'A' is marked on the smith chart as shown in figure. The normalized impedance of point A is –

$z_A = (1 – 0.65 j) = 1.2 \angle - 33.02^0$

$y_A = 1 / (1.2 \angle - 33.02^0) = 0.8 \angle 33.02^0 = (0.67 + 0.4 j) \Rightarrow 0.7 - 0.4 j$

(1) For the first path, $z_o \rightarrow z_A$; there is series capacitance

$\therefore 1 / (j\omega Z_oC) = z_A – z_o$

$= (1 – 0.65 j) - (1 + o j) = - 0.65 j$

$\therefore$ C = 1 / (2π x 900 10^6 x 50 x 0.65) = 5.44 PF.

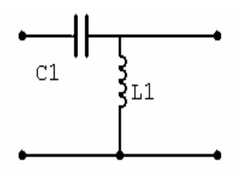
(2) For the second path, $z_A \rightarrow z_{opt}$; there is shunt inductance.

$\therefore Z_o / (j\omega L) = y_{opt} – y_A$

$= (0.7 – 0.4 j) – (0.7 + 0.4 j) = - 0.8 j$

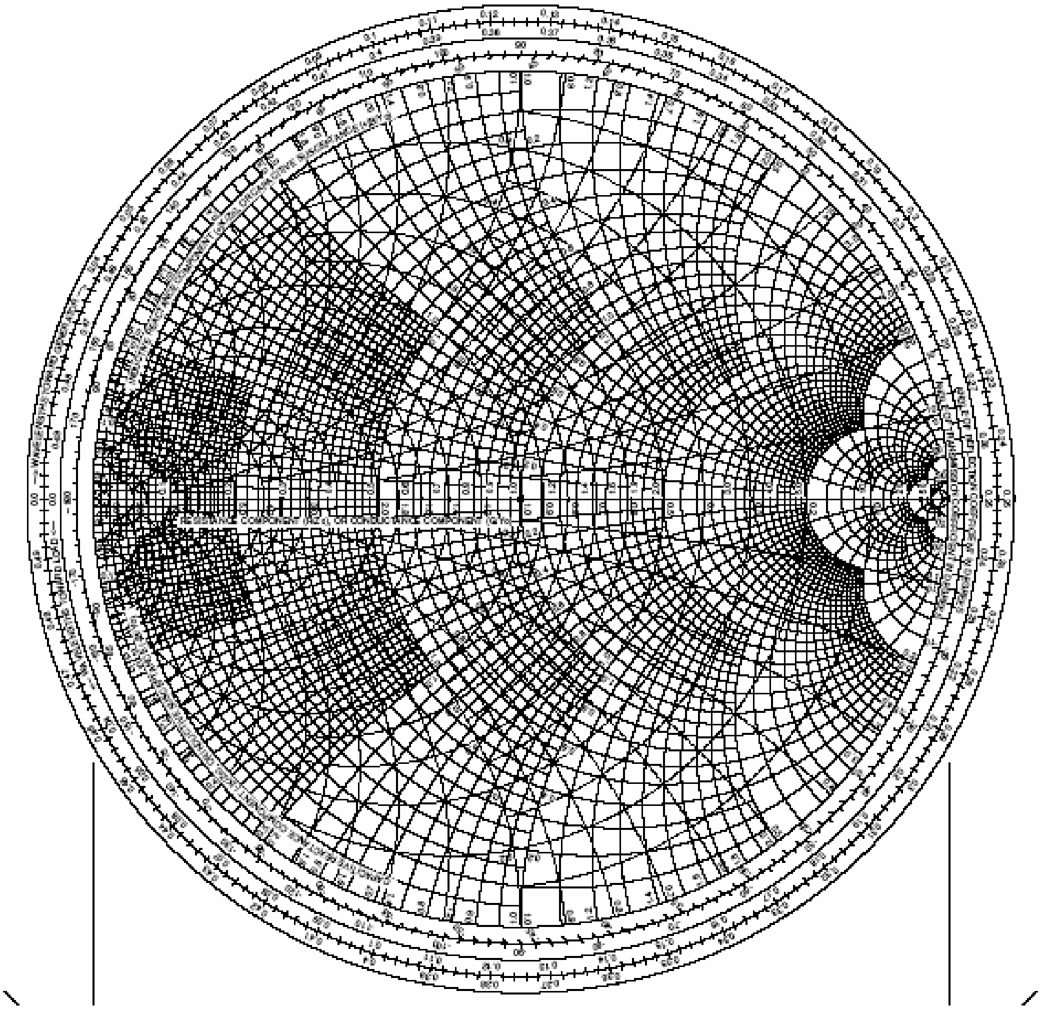
$\therefore$ L = Z_o / (ω x 0.8) = 50 / (2π x 900 10^6 x 0.8) = 11.05 nH

Thus the required input impedance matching network will be –



Where C1 = 5.44 PF

L1 = 11.05 nH



(2) Design of intermediate impedance matching network $Z_{out} \Rightarrow Z_{opt}$

For 900 MHz frequency, S_{22} is -

$S_{22} = (0.29 \angle - 83.8^0) = (0.03 - 0.29 j) \Rightarrow 0.03 - 0.3 j$

We have, Z_{out} given by the formula,

$$Z_{out} = Z_o [(1 + S_{22}) / (1 - S_{22})]$$

$$= 50 [(1 + (0.03 - 0.3 j)) / (1 – (0.03 - 0.3 j))]$$

$$= 50 [(1.03 - 0. 3 j)) / (0.97 + 0. 3 j))]$$

$$= 50 [(1.07 \angle -16.24^0) / (1.01 \angle 17.2^0)]$$

$$= 50 (1.06 \angle -33.44^0)$$

$$= 53 \angle -33.44^0 = (44.23 – 29.2 j)$$

Now, design of output impedance matching network, $Z_{out} \Rightarrow Z_{opt}$

i.e. $(44.23 – 29.2 j) \Rightarrow (55.26 + 34.28 j)$

The normalized impedances are –

We have,

$\mathcal{Z}_{opt} = 1.3 \angle 31.83^0 = 1.1 + 0.68 j \Rightarrow 1.1 + 0.7 j$

$y_{opt} = = 0.77 \angle - 31.81^0 = 0.65 - 0.4 j \Rightarrow 0.7 - 0.4 j$

$$\mathcal{Z}_{out} = Z_{out} / Z_o = (53 \angle -33.44^0) / 50$$

$$= 1.06 \angle - 33.44^0$$

$$= 0.9 - 0.58 j \Rightarrow 1 – 0.6 j$$

$$y_{out} = 1 / (1.06 \angle - 33.44^0)$$

$$= 0.94 \angle 33.44^0 = 0.78 + 0.51 j \Rightarrow 0.78 + 0.5 j$$

The points ‘A’ and ‘B’ are marked on the smith chart as shown in figure 4.12. The path is – $\mathcal{Z}_{out} \rightarrow \mathcal{Z}_A \rightarrow \mathcal{Z}_B \rightarrow \mathcal{Z}_{opt}$

The normalized impedances of point A and B are as –

$\mathcal{Z}_A = (0.8 + 0.6 j) = 1 \angle 36.86^0$

$y_A = 1 / (1 \angle 36.86^0) = 1 \angle - 36.86^0 = (0.8 - 0.6 j)$

$\mathcal{Z}_B = (0.8 – 0.7 j) = 1.06 \angle - 41.85^0$

$y_B = 1 / (1.06 \angle - 41.85^0) = 1 \angle 41.85^0 = 0.7 + 0.66 j$

(1) For the first path, $\mathcal{Z}_{out} \rightarrow \mathcal{Z}_A$; there is shunt inductance

$\therefore Z_o / (j\omega L) = y_A – y_{out}$

$$= (0.8 - 0.6 j) - (0.8 + 0.5 j) = - 1.1 j$$

$\therefore L = Z_o / (\omega \times 1.1) = 50 / (2\pi \times 900\ 10^6 \times 1.1) = 8.03$ nH

(2) For the second path, $z_A \rightarrow z_B$; there is series capacitance.

$\therefore 1 / (j\omega Z_o C) = z_B - z_A$

$$= (0.8 - 0.7 j) - (0.8 + 0.6 j) = -1.3 j$$

$\therefore\ C = 1 / (\omega Z_o \times 1.3) = 1 / (2\pi \times 900\ 10^6 \times 50 \times 1.3) = 2.72$ PF.

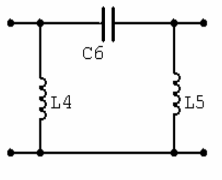
(3) For the third path, $z_B \rightarrow z_{opt}$; there is shunt inductance

$\therefore\ Z_o / (j\omega L) = y_{opt} - y_B$

$$= (0.7 - 0.4 j) - (0.7 + 0.66 j) = - 1.04 j$$

$\therefore L = Z_o / (\omega \times 1) = 50 / (2\pi \times 900\ 10^6 \times 1) = 8.84$ nH

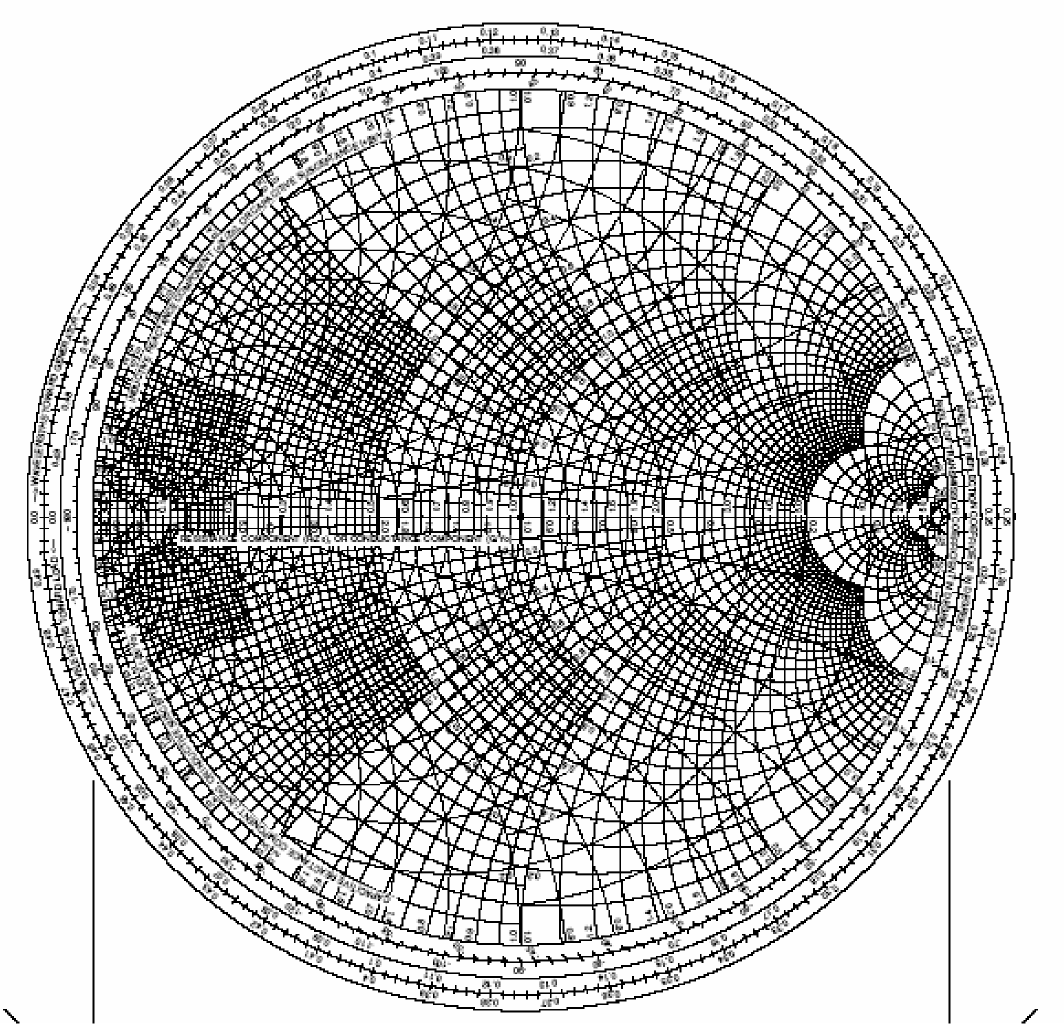
Thus the required intermediate impedance matching network will be –



Where L4 = 8 nH

L5 = 8.84 nH

C6 = 2.72 PF



(3) Design of output impedance matching network $Z_{out} \Rightarrow Z_o$

We have,

z_{out} = $1.06 \angle -33.44^0 = 0.9 - 0.58\,j \Rightarrow 1 - 0.6\,j$

y_{out} = $0.94 \angle 33.44^0 = 0.78 + 0.51\,j \Rightarrow 0.8 + 0.5\,j$

$z_o = Z_o / Z_o = 50 / 50 = (1 + o\,j)$

$y_o = 1 / (1 + 0\,j) = 1 \angle 0^0 = (1 + 0\,j)$

The point 'A' is marked on the smith chart as shown in figure 4.13. The path is –

$z_{out} \rightarrow z_A \rightarrow z_o$

The normalized impedance of point A is –

$z_A = (1 + 0.5\,j) = 1.1 \angle 26.56^0$

$y_A = 1 / (1.1 \angle 26.56^0) = 0.9 \angle -26.56^0 = (0.8 - 0.4\,j)$

(1) For the first path, $z_{out} \rightarrow z_A$; there is shunt inductance

$\therefore\; Z_o / (j\omega L) = y_A - y_{out}$

$= (0.8 - 0.4\,j) - (0.8 + 0.5\,j) = -0.9\,j$

$\therefore L = Z_o / (\omega \times 0.9) = 50 / (2\pi \times 900\; 10^6 \times 0.9) = 9.82\text{ nH} \Rightarrow 10\text{ nH}$

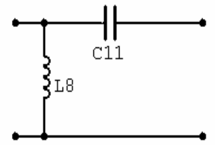
(2) For the second path, $z_A \rightarrow z_o$; there is series capacitance.

$\therefore 1 / (j\omega Z_o C) = z_o - z_A$

$= (1 + 0\,j) - (1 + 0.5\,j) = -0.5\,j$

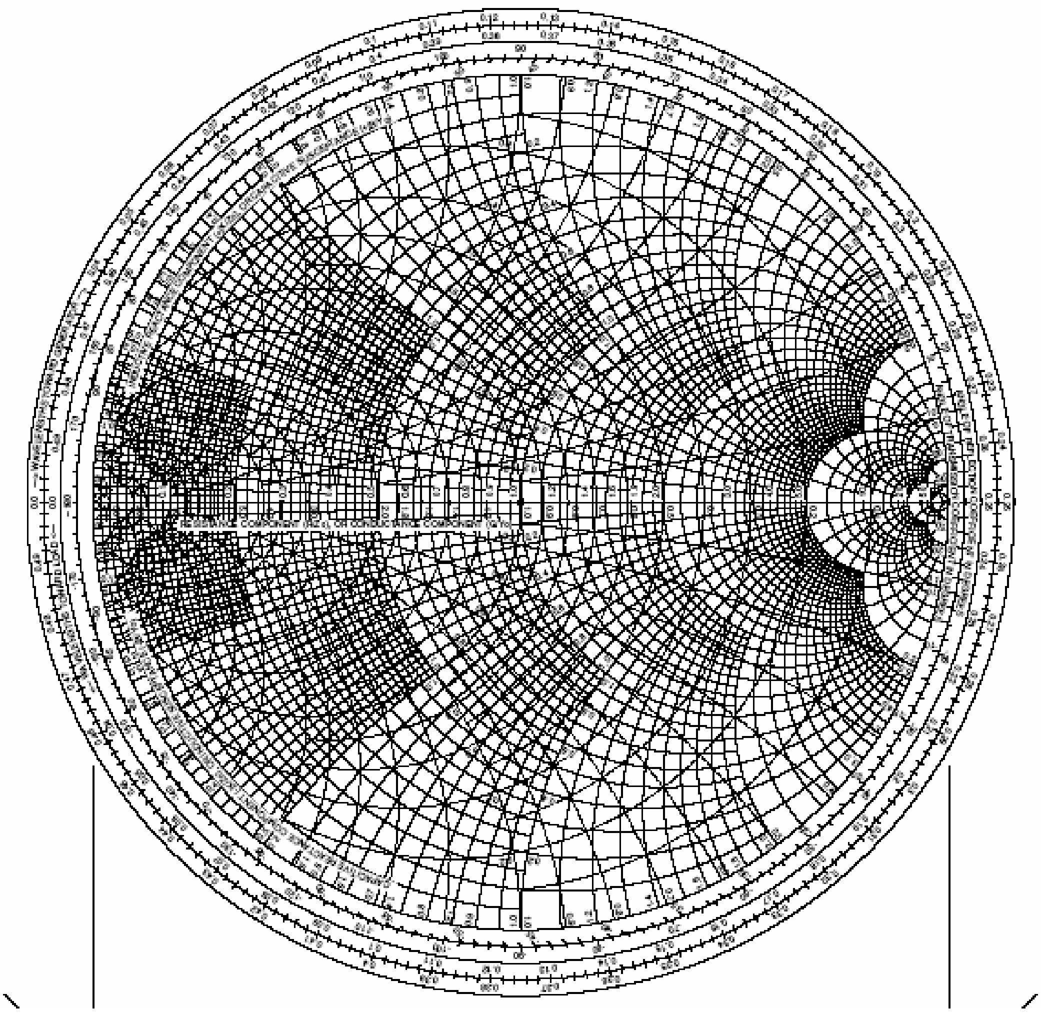
$\therefore\; C = 1 / (\omega Z_o \times 0.5) = 1 / (2\pi \times 900\; 10^6 \times 50 \times 0.5) = 7.07\text{ PF} \Rightarrow 7\text{ PF}$

Thus the required output impedance matching network will be –



Where L8 = 9.82 nH

C11 = 7.07 PF



4.3 Stability calculations

Aside from low noise figure and high gain, an LNA must be stable. When an amplifier is unconditionally stable, no oscillations will occur no matter what source and load reflection coefficients Γ_S and Γ_L are presented. Input and output reflection coefficients Γ_{IN} and Γ_{OUT} respectively depend on load and source reflection coefficients Γ_L and Γ_S as follows -

$\Gamma_{IN} = S_{11} + [(S_{12}S_{21}\Gamma_L) / (1 - S_{22}\Gamma_L)]$ -- (4.1)

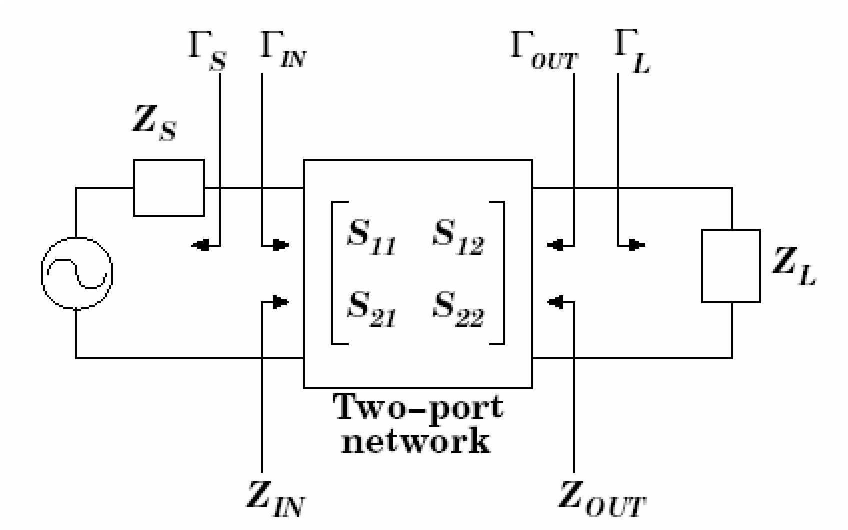


Figure 4.14 - Two port network and associated reflection coefficients.

$\Gamma_{OUT} = S_{22} + [(S_{12}S_{21}\Gamma_S) / (1 - S_{11}\Gamma_S)]$ --- (4.2)

where all reflection coefficients are as shown in Figure 4.14.

Oscillation occurs when there is positive feedback at the input or output of the amplifier, which is also represented as an input or output reflection coefficient with a magnitude greater than one ($|\Gamma_{IN}| > 1$ or $|\Gamma_{OUT}| > 1$).

A check for unconditional stability can be made when the S-parameters of an amplifier are known. Stability factor k is defined as the following :

$k = [1 + |\Delta|^2 - |S_{11}|^2 - |S_{22}|^2] / [2 |S_{12}| |S_{21}|]$ ---------------------------- (4.3)

where $\Delta = S_{11}S_{22} - S_{12}S_{21}$

.

For unconditional stability,

$$k > 1.$$

and

$$|\Delta| < 1$$

At 700 MHz, the ATF-54143 has the following measured S-parameters

$$S_{11} = 0.76 \angle -98.8^0$$

$$S_{12} = 0.04 \angle 47.3^0$$

$$S_{21} = 17.2 \angle 117.2^0$$

$$S_{22} = 0.35 \angle -71.3^0$$

Substituting the above S-parameters into above Equations of k and Δ, results in a stability factor k of 0.37 and $|\Delta| = 0.46$, which means the ATF-54143 is not unconditionally stable and that there are possible values of Γ_S and / or Γ_L that will cause $|\Gamma_{OUT}|$ and/or $|\Gamma_{IN}|$ to be greater than 1. These values can be determined by investigating the areas of stability on the Smith Chart in the Γ_S-plane and Γ_L-plane.

Areas of stability on the Smith Chart can be determined with stability circles, circles where values of Γ_S and Γ_L respectively cause $|\Gamma_{OUT}|$ and $|\Gamma_{IN}|$ to be equal to 1. The Smith Chart itself encompasses an area in the Γ-plane where $|\Gamma| \leq 1$. Since the transistor is not unconditionally stable, there will be areas on the Smith Chart that overlap with areas of the stability circle in the Γ_S-plane and/or Γ_L-plane. If $|S_{11}| < 1$ and $|S_{22}| < 1$, the stable regions are the regions that contain $\Gamma_S = 0$ and $\Gamma_L = 0$ (the centers of each Smith Chart) and are bounded by the Smith Chart boundary and stability circle. The following are the equations for the radius r_L and center C_L of the output stability circle. where values of Γ_L cause $|\Gamma_{IN}|$ to be equal to 1.

$$r_L = |\, [\, S_{12}S_{21} / (\, |S_{22}|^2 - |\Delta|^2\,)\,] \text{ -- } (4.4)$$

$$C_L = [(S_{22} - \Delta S^*_{11})^* / (| S_{22} |^2 - | \Delta |^2)] \text{ ------------------------------ } (4.5)$$

Similarly for the Γ_S-plane, the radius r_S and center C_S of the input stability circle are defined as follows

$$r_S = | [S_{12}S_{21} / (| S_{11} |^2 - | \Delta |^2)] \text{ --- } (4.6)$$

$$C_S = [(S_{11} - \Delta S^*_{22})^* / (| S_{11} |^2 - | \Delta |^2)] \text{ ------------------------------ } (4.7)$$

Figure 4.15 demonstrates how the overlap of the stability circles with the Smith Chart indicates unstable regions of the Smith Chart if both $|S_{11}|$ and $|S_{22}|$ are less than 1. When an amplifier is unconditionally stable, the stability circles lie completely outside the Smith Chart area.

Using the S-parameters of the ATF-5143 at 700 MHz, its stability circles can be calculated and mapped out (see figure 4.16) with radii and centers :

$r_L = 7.67$

$C_L = 7.33 \angle -89.13^0$

$r_S = 1.86$

$C_S = 2.4 \angle 104.97^0$

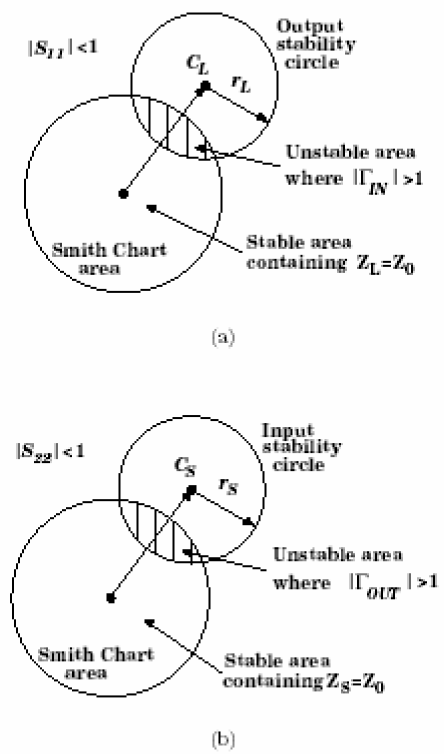


Figure 4.15 - Stability circles and unstable regions in the Smith Chart. a) The output stability circle in the Γ_L-plane. b) The input stability circle in the Γ_S-plane.

The area where values of Γ_S cause transistor noise to be lowest (calculated Γ_{opt} values) is stable and therefore designing an LNA for lowest noise is viable. Additional measures to improve stability can be taken. Adding resistance to the drain of a transistor as well as a small shunt capacitance can provide stability at higher frequencies. Extra inductance added in the source leads of a transistor also improves stability while improving input return loss.

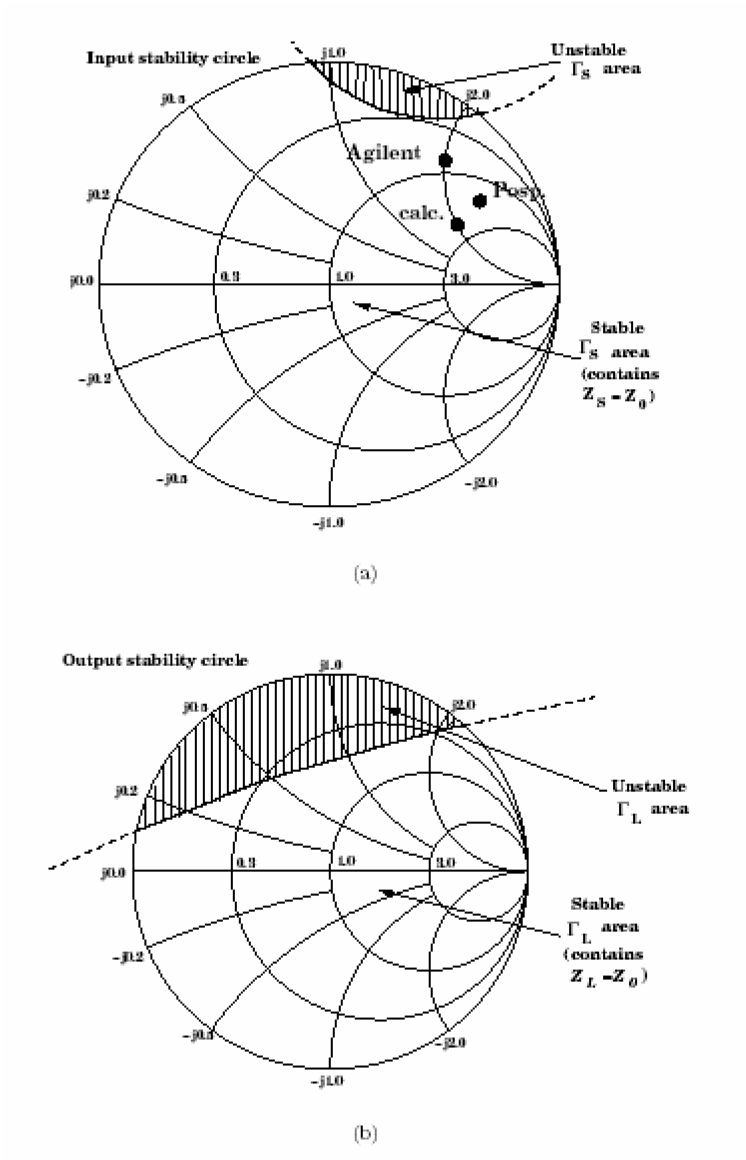


Figure 4.16 - Stability circles of the ATF-54143 at 700 MHz. a) The input stability circle. b) The output stability circle.

4.4 Design of microstrip transmission line for 50 Ω characteristic impedance

In microstrip line, the grounded metallization surface covers only one side of the dielectric substrate, as shown in figure. In this case, the electric & magnetic field lines are located in both the dielectric region between the strip conductor & the ground plane and in the air region above the substrate. As a result, the electromagnetic wave propogated along a microstrip line is not a pure Tem,since the phase velocities in these two regions are not the sameBut in a quasistatic approximation, which gives sufficiently accurate results as long as the height of the dielectric substrate is very small compared with the wavelength, it is possible to obtain the analytical expressions for the electrical characteristics.

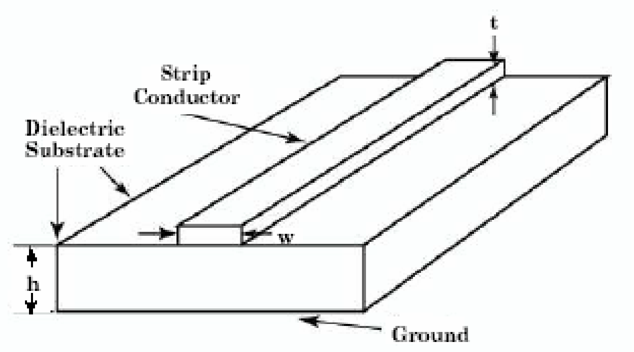


Figure 4.17 – Microsrip Geometry.

We have chosen a substrate – Ultralam 2000. It is made up of PTFE (polytetra-fluoroehylene). PTFE is thermally most stable & chemically most resistant carbonaceous compound. It is unaffected by sunlight, moisture, and virtually all chemicals. Electrical properties are very constant over temperature and a wide range of frequencies.

Some of the properties of Ultralam 2000 from the data sheet are-

Sr. No.	Name of parameters	Value
1	Dielectric constant (εr)	2.5
2	Loss tangent	0.0022
3	Resistivity	4.1 e^7
4	Metal thickness	0.035 mm

5	Metal Roughness	2.388 e^{-3} mm
6	Substrate height (h)	0.762 mm

Table 4.1 Parameters of Ultralam 2000 substrate

From the datasheet of Ultralam 2000, a product by Roger's Corporation, we have the following parameters for the substrate,

Dielectric constant = εr = 2.5

Substrate height = h = 0.762 mm

Characteristic impedance = 50 Ω

If 'W' is the width of the microstrip and h is the height of the microstrip then W/h ratio is given by the formula as-

$$W/h = (8 \times e^{A}) / (e^{2A} - 2) \quad \text{--} \quad (4.8)$$

Where the factor A is given by,

$$A = 2 \pi (Zo/Z_f) [(\varepsilon r +1)/2]^{1/2} + [(\varepsilon r -1)/(\varepsilon r + 1)] (0.23 + 0.11/ \varepsilon r) \text{ ----- } (4.9)$$

Here $Z_f = (\mu o / \varepsilon o) = 376.8\ \Omega$

By putting the values in the above expression of A, we get the value of A as,

A = 1.22

Putting the value of A in above equation (1), we get W/h ratio as,

W/h = 2.862

Therefore W = 2.862 x h = 2.862 x 0.762 mm

W = 2.18 mm ≅ 2.2 mm

This is the width of the microstrip for 50 Ω characteristic impedance

We can compute the characteristic impedance of the line to verify our result, we have,

$$Zo = Z_f / [(\varepsilon_{eff})^{1/2} (1.393 + (W/h) + (2/3) \ln (W/h + 1.444))] \text{ -------------- } (4.10)$$

Where

$\varepsilon_{eff} = [(\varepsilon r + 1) / 2] + [(\varepsilon r - 1) / 2] (1 + 12 h/W)^{-1/2}$ ------------------------ (4.11)

Putting the values $\varepsilon r = 2.5$ and $h/W = 1/ 2.862$ in above equation , we have

$\varepsilon_{eff} = 2.08$

Putting the value of $\varepsilon_{eff} = 2.08$, $W/h = 2.862$, $Z_f = 376.8\ \Omega$ in equation (3) we get,

$Zo = 49.97\ \Omega \cong 50\ \Omega$.

Hence verified.

Thus for microstrip width W = 2.2 mm, we can get the characteristic impedance of 50 Ω.

4.5 Design of Inductors :

The inductor of value L nH (Air-wound coil Inductor) is given by,

$L = B^2 n^2 / 0.45 B + A$ -- (4.12)

Where-

L = Coil inductance

B = Average coil diameter (mm)

n = number of turns

A = Coil length (mm)

These inductors are used in RF circuits (For frequency 10 MHz – 1 GHz) to design filters, impedance, matching circuits & RF blocking components.

4.6 GENESYS 2005 – A simulation software for RF & Microwave Amplifier Design

In design of broad-band high frequency transistor amplifiers is a difficult and challenging problem in active network theory of great practical importance. The difficulties are largely due to the number of requirements at a time. To set one of the parameter by changing the values of the appropriate circuit elements, another parameter gets affected. Therefore in broad-band high frequency transistor amplifiers, there is need of optimization of the component values to get optimum result. At present, a rigorous design theory for broad-band transistor amplifiers is not available and designers must rely on computer-aided and experimental cut and try techniques which are often inadequate.

GENESYS™ is an integrated electronic design automation (EDA) software suite for radio frequency (RF) and microwave design. It is a powerful tool that helps you design better products in less time.

GENESYS provides state-of-the-art performance in a single easy-to-use design environment that is fast, powerful, and accurate. It improves the way your design teams work together and the way you work within a manufacturing environment.

This version of GENESYS represents a dramatic paradigm shift that adds a number of new features to help improve your productivity. It offers a totally new design concept that changes the way you think about designs. GENESYS 2005 also provides a more efficient way for you to do things.

New GENESYS Features – New features include a redesigned, easy-to-use user interface, new and enhanced SPECTRASYS models, new object libraries, and full XML support.

New Paradigm for Designs, Analyses, and Graphs – Major changes in the way you create or use designs, analyses, and graphs save time and allow greater flexibility and control.

The Eagleware support forums provide a convenient way to exchange information with other Eagleware users. Search the forums for an answer to a question about GENESYS or post a new question and receive help from Eagleware staff and other users. You'll find that most of your questions are already answered.

The forums address a variety of topics including design issues, product questions, and installation questions. We even have a forum for ideas that you think would be great in future versions of GENESYS.

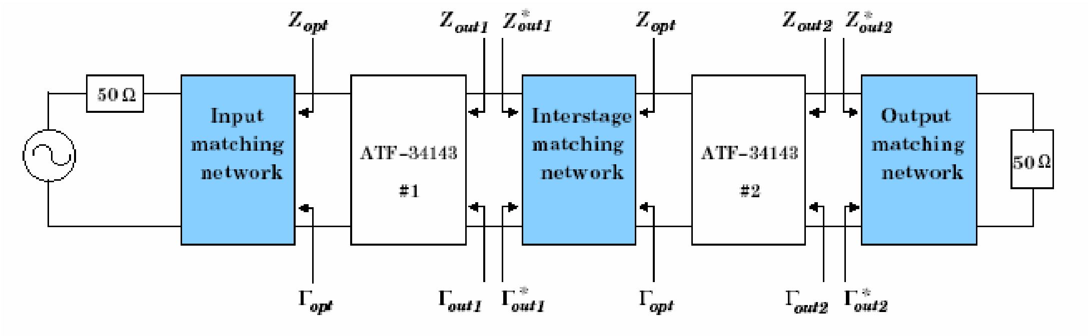
You can access the forums at the Eagleware Web site (www.eagleware.com). Linear simulation calculates S-parameters and noise parameters of a schematic design. If the circuit contains nonlinear elements, DC simulation is first run automatically and the nonlinear devices are linearized at the DC operating point.

For broadband design, presenting the correct impedance to the device across the band and a flat gain requires balancing multiple goals. This is best accomplished using a modern simulator such as GENESYS to optimize all of the requirements simultaneously. For even broader bandwidth, the MATCH synthesis program can be used to find a network which presents near optimum impedance to the device over the entire band.

The input network has been optimized so that at the middle of the frequency band a 50 ohm source will provide the optimum noise performance. This is verified by examining the noise figure versus frequency plot on the left in the SUPERSTAR output screen. The gain flatness was acheived by optimization of the output matching network. Better output return loss could have been acheived by optimizing for match instead of gain flatness. The match at the input falls where it must because the input network is optimized for best noise and not best match.

The layout below was generated for the completed amplifier. The microstrip lines and discontinuities were automatically generated in LAYOUT

4.7 Impedance matching networks for LNA using ATF 54143 after simulation



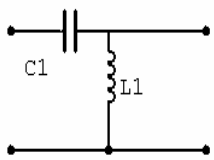
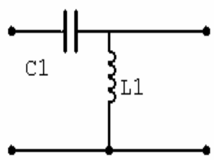
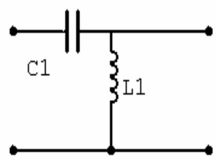
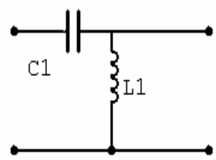
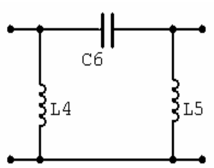
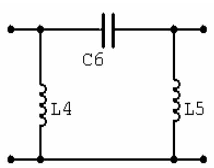
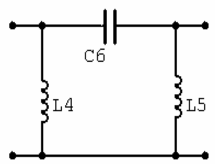
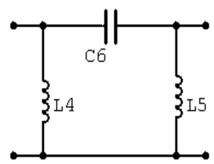
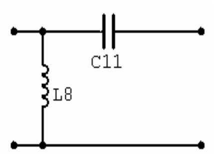
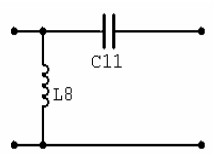
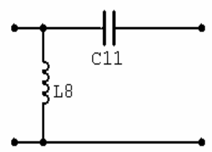
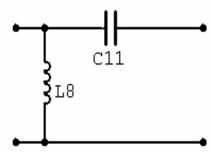
Frequency → / Impedance Matching n/w ↓	500 MHz (Theoretical Design)	700 MHz (Theoretical Design)	900 MHz (Theoretical Design)	Simulation Genesys 2005
Input Impedance Matching Network Zo ⇒ Zopt	 C1 = 6.36 PF L1 = 19.89 nH	 C1 = 5.68 PF L1 = 14.21 nH	 C1 = 5.44 PF L1 = 17.05 nH	 C1 = 15 PF L1 = 33 nH
Intermediate Impedance Matching Network Zout ⇒ Zopt	 L4 = 17.68 nH L5 = 20.4 nH C6 = 3.54 PF	 L4 = 8.74 nH L5 = 10.33 nH C6 = 3 PF	 L4 = 8 nH L5 = 8.84 nH C6 = 2.72 PF	 L4 = 22 nH L5 = 47 nH C6 = 8.2 PF
Output Impedance Matching Network Zout ⇒ Zo	 L8 = 15.9 nH C11 = 6.36 PF	 L8 = 12 nH C11 = 7 PF	 L8 = 9.82 nH C11 = 7 PF	 L8 = 18 nH C11 = 10 PF

Figure 4.18 - Impedance matching networks for LNA using ATF 54143 after simulation – a comparative chart.

4.8 Circuit Diagram of LNA with component values obtained from simulation

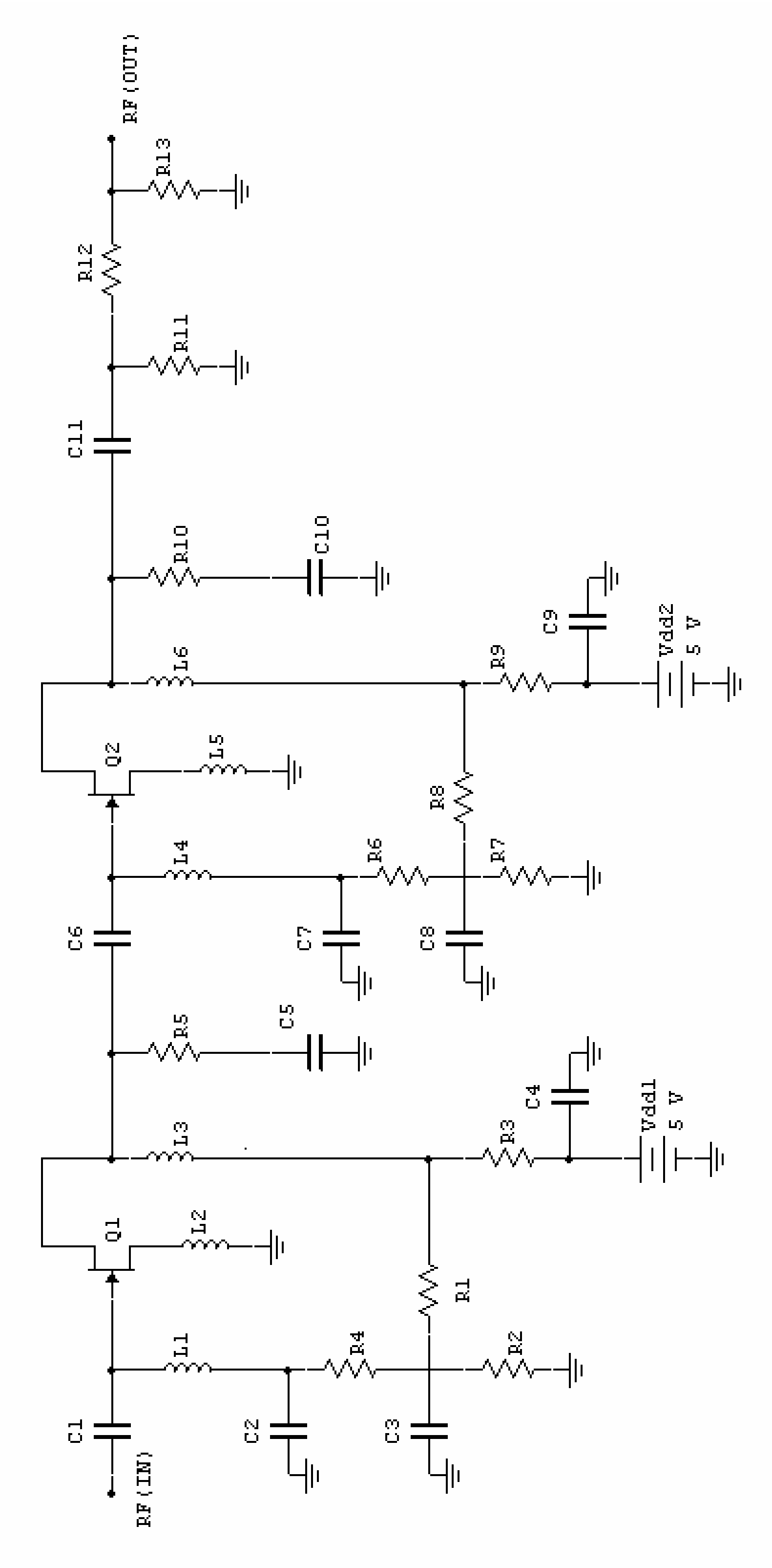


Figure 4.19 - Circuit Diagram of LNA with component values obtained from simulation

Component Value list (simulation)

Component	Value
C_1	14.2 PF
L_1	30.81 nH
C_2	13.8 PF
C_3, C_4	10 nF
C_8, C_9	10 nF
R_1, R_8	295 Ω
R_2, R_7	1205 Ω
R_3, R_9	32.3 Ω
R_4, R_6	56 Ω
L_2, L_5	1 nH
L_3	20.36 nH
C_5, C_{10}	10 PF
C_6	7.6 PF
L_4	46.4 nH
C_7	13.7 PF
L_6	19.2 nH
C_{11}	10 PF
R_{11}, R_{13}	150 Ω
R_{12}	39 Ω
Q_1, Q_2	ATF 54143
V_{dd1}, V_{dd2} DC Voltage	+ 5 V DC

Table 4.2 - Component value (obtained by simulation) list of LNA

4.9 Optimization of the component values for better performance

Component	S (1,1) Input return loss	S (2,2) Output return loss	S (2,1) Gain	Noise factor (NF)	Stability factor (K)
C_1↑ From optimum value	Graph on negative side goes away from x-axis	No effect	No effect	NF gets increased for optimum value	No effect
L_1↑	Graph on negative side	No effect	Graph on positive side goes	NF gets increased for	Gets decreased

	approaches to x-axis		away from x-axis	optimum value	
C2↑	Graph on negative side approaches to x-axis	No effect	Graph on positive side goes away from x-axis	NF gets decreased for optimum value	No effect
C3↑	No effect	No effect	No effect	No effect	No effect
L2↑	For optimum value it gives better result	No effect	Graph on positive side goes away from x-axis	For optimum value it gives better result	Gets increased
L3↑	For optimum value it gives better result	No effect	For optimum value it gives better result	For optimum value it gives better result	No effect
C4↑	No effect	No effect	No effect	No effect	No effect
C5↑	No effect	No effect	No effect	No effect	No effect
C6↑	Graph on negative side approaches to x-axis	No effect	No effect	No effect	No effect
L4↑	No effect	No effect	No effect	No effect	No effect
C7↑	Graph on negative side approaches to x-axis	Graph on negative side approaches to x-axis	Gets decreased	No effect	Gets increased
C8↑	No effect	No effect	No effect	No effect	No effect
L5↑	No effect	For optimum value it gives better result	For optimum value it gives better result	No effect	Gets increased
L6↑	No effect	Graph on negative side goes away from x-axis	Gets increased	No effect	Gets decreased
C9↑	No effect	No effect	No effect	No effect	No effect
C10↑	No effect	No effect	No effect	No effect	No effect
C11↑	No effect	No effect	No effect	No effect	No effect
R11, R12, R13	No effect	No effect	Gets increased	No effect	No effect

Table 4.3 - Optimization of the component values for better performance

4.10 Graphs of LNA parameters obtained by theoretical & simulation

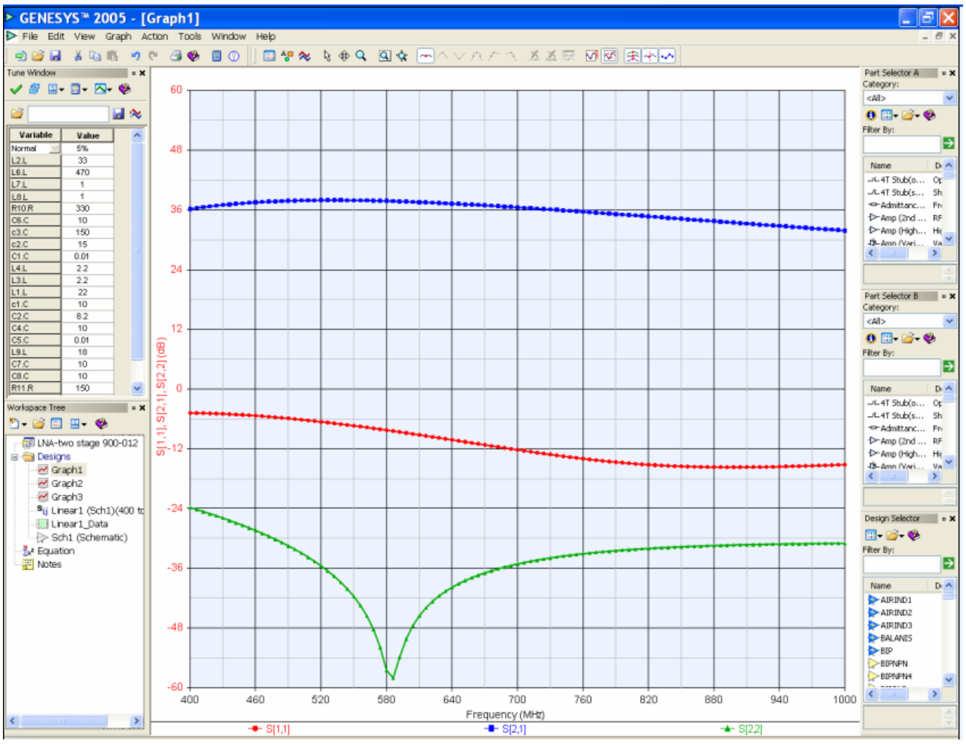


Figure 4.20 - Graph of Gain (S21-Blue), Input return loss (S11-Red), output return loss (S22 –Green)

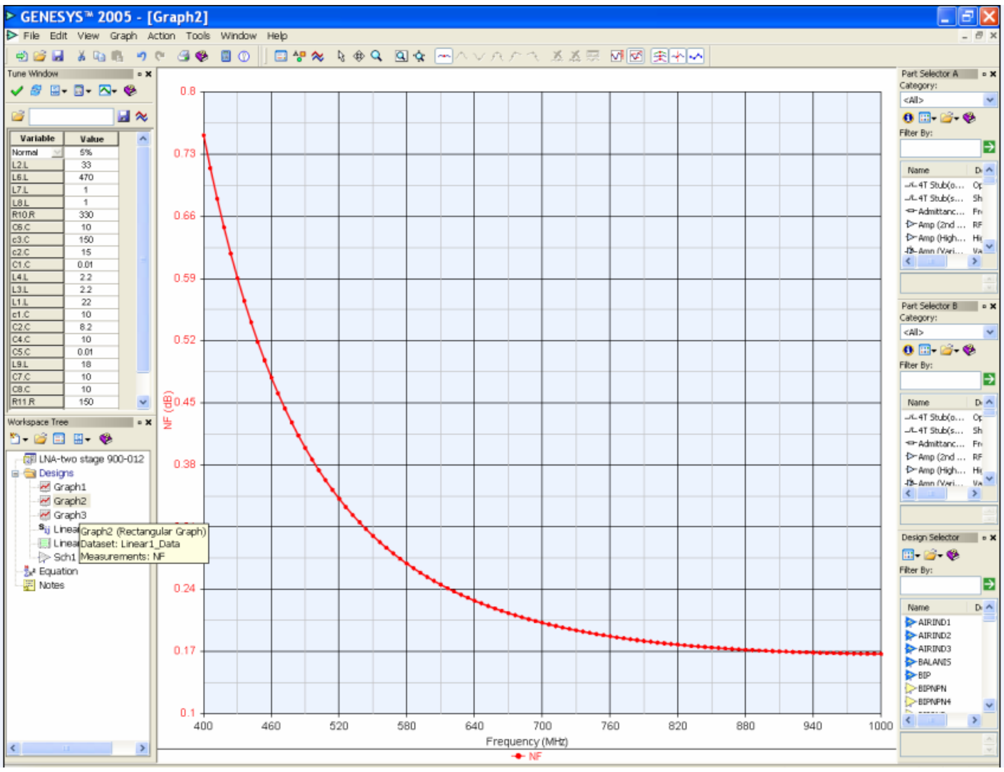


Figure 4.21 - Graph of noise figure (NF)

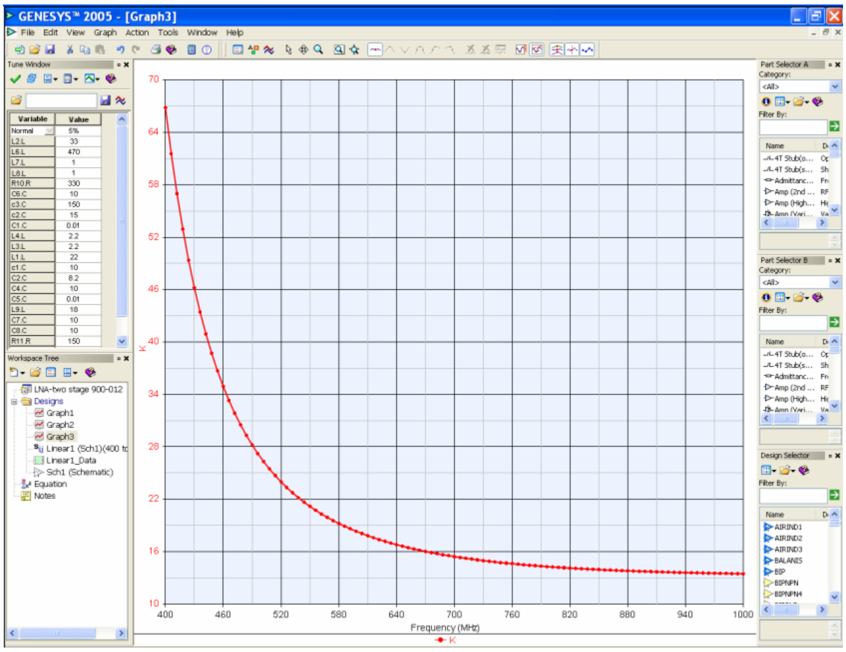


Figure 4.22 Graph of Stability factor (K)

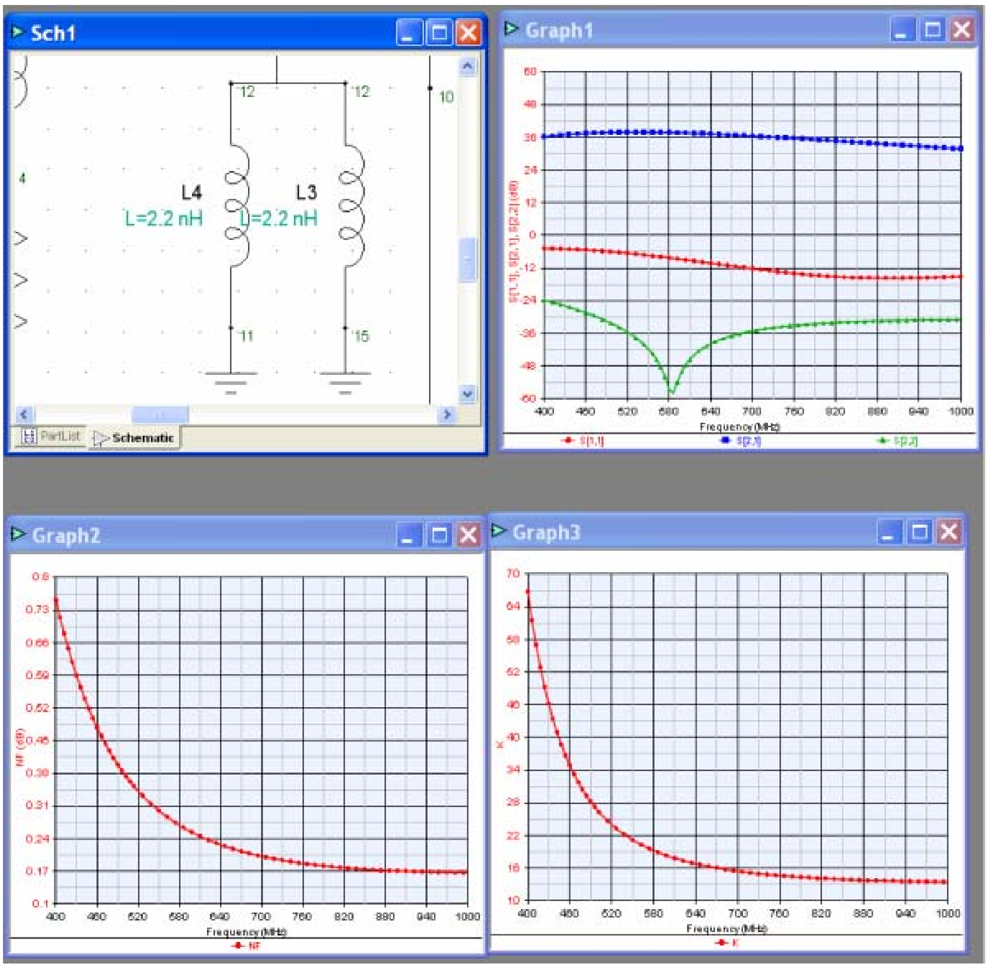


Figure 4.23 - Graphs of LNA parameters using theoretical & simulated design

4.11 LNA circuit Diagram with standard component value (Actual circuit) :

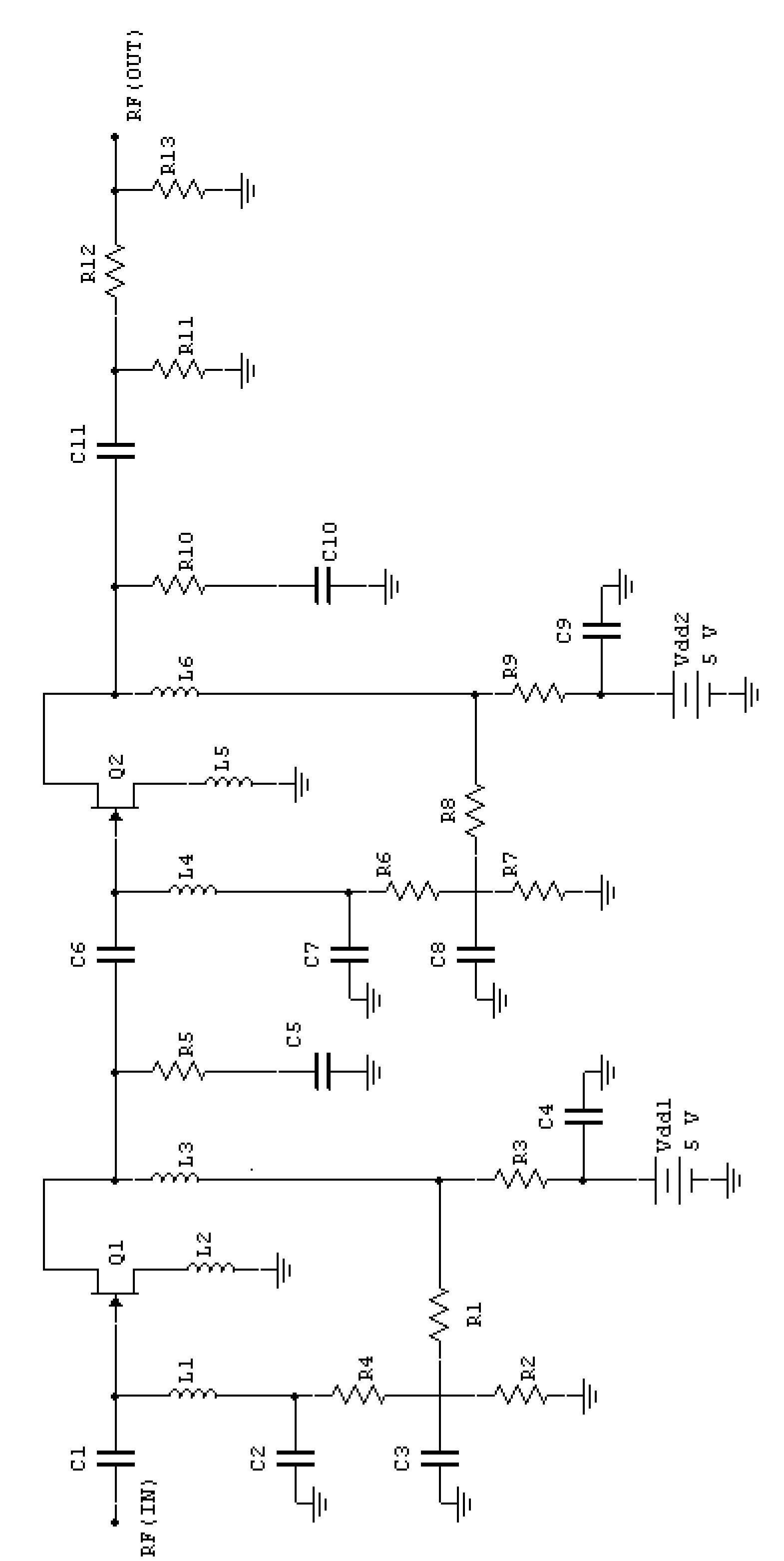


Figure 4.24 - LNA circuit Diagram with standard component value (Actual circuit)

4.12 Component value / function List : (Table 4.4 – Component value & Function)

Component	Type	Value	Function
C1	CC1608(0603) Chip capacitor	15 PF	Impedance matching
L1	LM 250 molded	33 nH	Impedance matching
C2	CC1608(0603)	15 PF	Provide a low impedance inband RF bypass for the matching network
C3, C4	CC1608(0603)	10 nH	Provide the low frequency RF bypass for resistors R4, R3 and R6 , R9.There values should be chosen carefully as they also provide a termination for low frequency mixing products. These mixing products are as a result of two or more inband signals mixing & producing third order inband distortion products. The low frequency. The low frequency products are bypassed by these capacitors. For best suppression of third order distortion products their value should be 0.01 μF (10 nF). Smaller values of capacitor will not suppress the generation.
C8, C9	CC1608(0603)	10 nH	
R1, R8	RC1608(0603)	295 Ω	Biasing resistors
R2, R7	RC1608(0603)	1205 Ω	Biasing resistors
R3, R9	RC1608(0603)	32.3 Ω	Provide low frequency termination for the device. The resistive termination improves low frequency stability
R4, R6	RC1608(0603)	56 Ω	
L2, L5	Source Inductance (msind)	1 nH	Increases stability of the network
L3	LM 250 molded	22 nH	Impedance matching
C5, C10	CC1608(0603)	10 PF	Provides stability to device.
C6	CC1608(0603)	8.2 PF	Impedance matching
L4	LM 250 molded	47 nH	Impedance matching
C7	CC1608(0603)	15 PF	Provide a low impedance inband RF bypass for the matching network
L6	LM 250 molded	18 nH	Impedance matching
C11	CC1608(0603)	10 PF	Impedance matching
R11, R13	RC1608(0603)	150 Ω	Attenuator
R12	RC1608(0603)	39 Ω	Attenuator
Q1, Q2	ATF 54143 (SOT 343)	pHEMT	Pseudomorphic HEMT, used for microwave frequencies due to low noise temperature, better stability and gain.
Vdd1, Vdd2 DC Voltage	Signal, Coplanar	+ 5 V DC	DC voltage supply for biasing network.

4.13 Layout design of LNA :

After finalization of the standard component values, microstrips are added between each components of the schematic with the help of Genesys 2005 software. Microstrips are added to get 50 ohm characteristic impedance throughout the circuit connections. It is essential for low noise figure of the device.

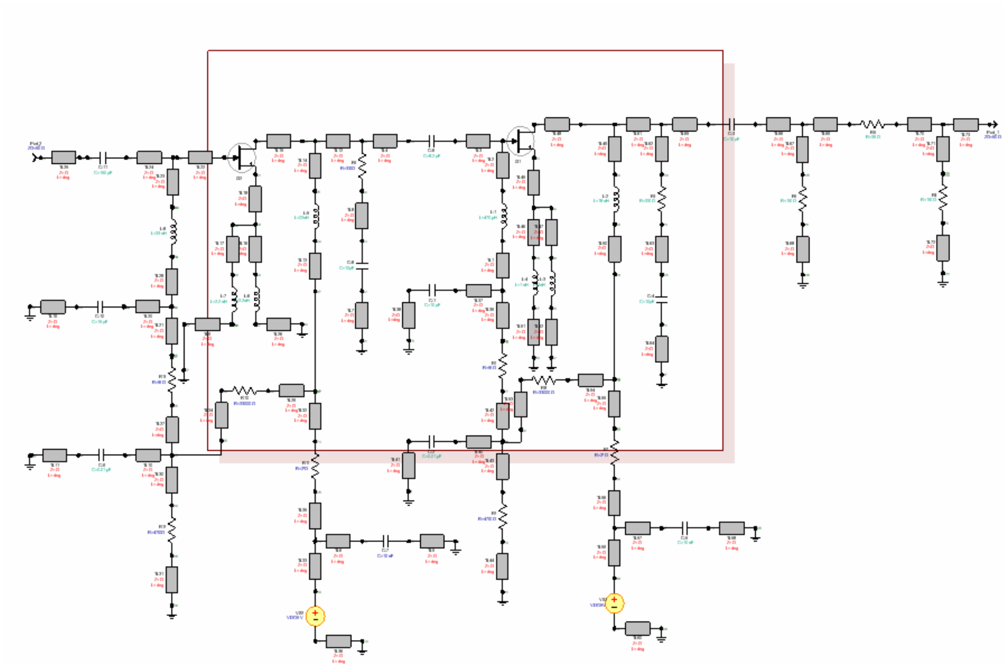


Figure 4.25 – LNA schematic with microsrip lines.

After connecting the microsrips of width 2 mm and appropriate length varies from 2mm – 5 mm , Layout of the LNA is designed using Genesys 2005 – simulation software. The layout obtained is –

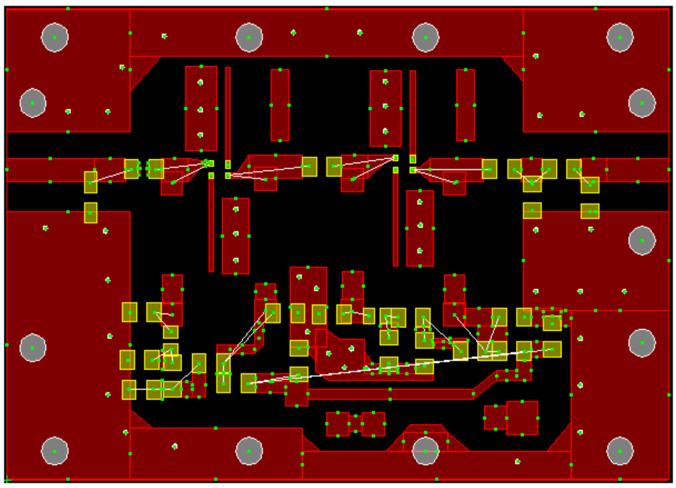


Figure 4.26 - Layout of LNA

4.14 Practical procedure to build LNA Hardware :

(1) After Layout (in the appropriate size- 75 mm x 50mm) obtained from Genesys 2005, the negative print of layout is taken on transparency. By taking PCB of 75 mm x 50mm, the print is pressed with the help of iron to heat the PCB so that there is black layer on it, indicating layout. Make sure that PCB black layer is correctly matched with layout.

(2) Now, PCB is inserted into Ferrichloride solution. The exposed copper part of PCB (other than layout) react with the solution and get dissolved in it. Now, we get the PCB ready for the connection after verification and cleaning with polish paper or steel wool.

(3) Mount the lumped component at appropriate places with tinning. Tin is used for soldering the components with solder gun (Temperature control- Weller's)

(4) Fit the PCB into Chassis using screws. The dimensions of Chassis and PCB are considered at the design stage.

(5) Connect the SMA connectors to Chassis and PCB for RF IN and RF OUT.

4.15 Noise figure and Gain measurement using Noise Figure Analyzer (NFA) :

Figure 4.27 - NFA (Agilent- N8974A) with a Normal Noise Source connected.

To measure noise figure and gain of LNA for different frequencies, by Noise Figure Analyzer (NFA) following steps are involved –

(1) Make instrument 'ON' by using key on the instrument. Wait 60 min. for warm up time.
(2) Select 'Frequency / points' keys and select frequency range and points (start frequency, stop frequency, Center frequency)
(3) Connect noise source. You may need to use connector adaptors to connect the noise source output to the NFA input during calibration. The connectors you use need to be included in the measurement.

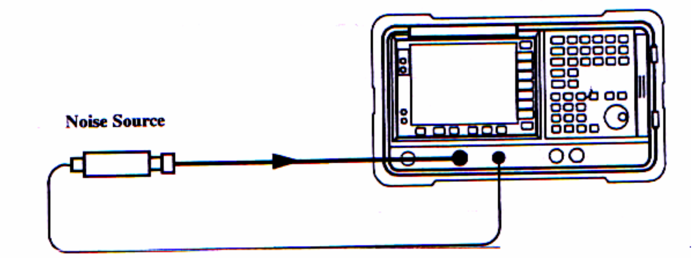


Figure 4.28 - NFA with a Normal Noise Source connected

(4) Press 'ENR' key. Go to ENR table. Edit the ENR table for the noise source.
(5) Press 'Result'. From NF, Gain, Teffective, Y factor; select any two parameters for which you have to study the graphs.
(6) Press 'Marker' button. Select four markers one by one and allocate frequency to each marker for which you have to see the parameters.
(7) Press 'Calibrate' for two times.

(8) Check the calibrations-‘Corr’. Values of marker should be nearly zero.
(9) Now, connect noise source along with DUT (LNA) and give DC supply to DUT.

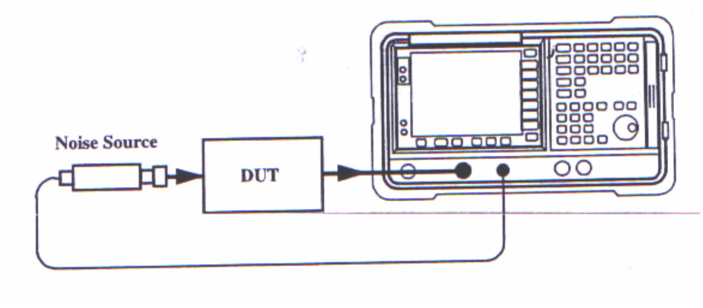
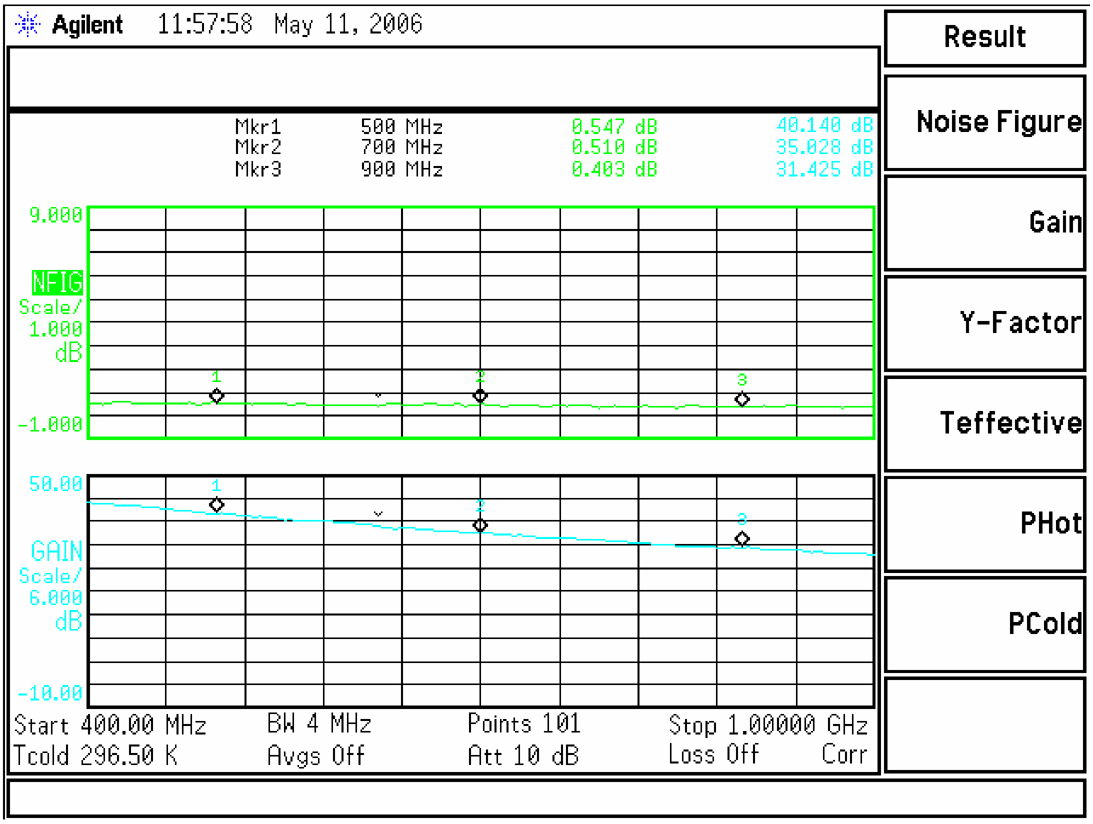


Figure 4.29 - NFA with a DUT and Noise Source connected to measure NF, Gain

(10) Graphs will be displayed on the screen.
(11) Save or print the graphs.

4.16 Graphs of LNA obtained with the noise figure Analyzer



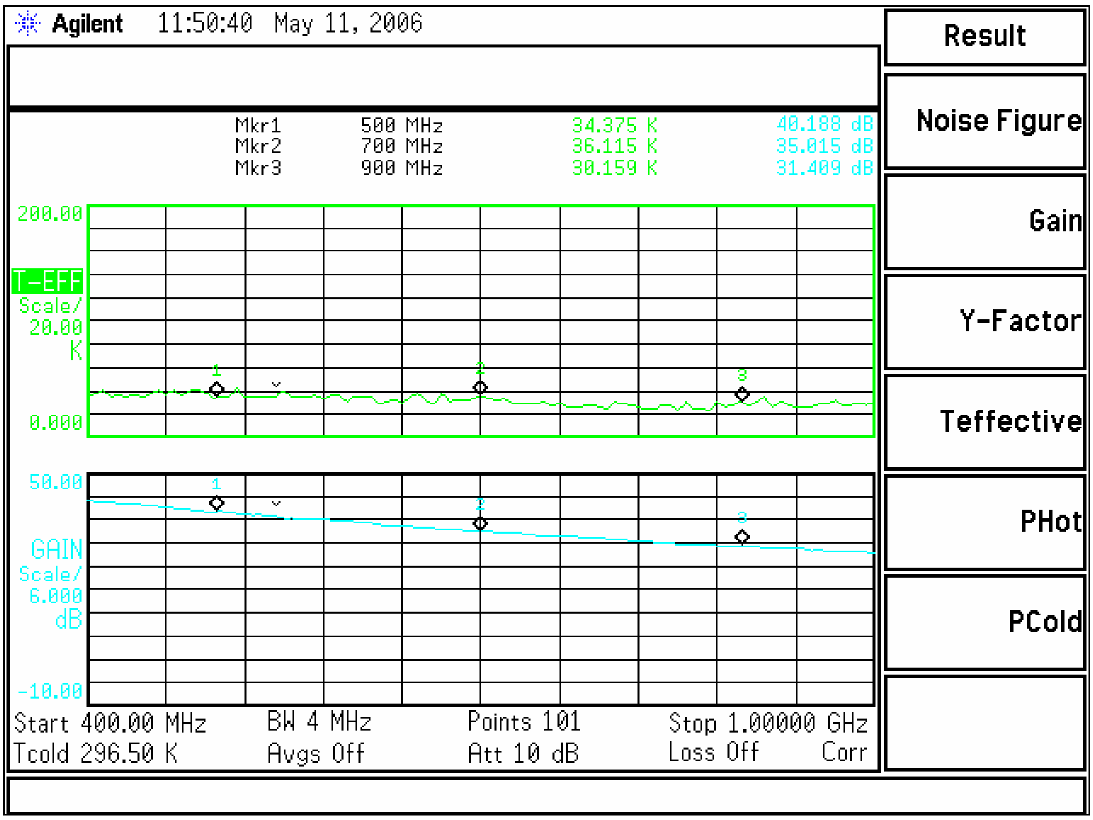
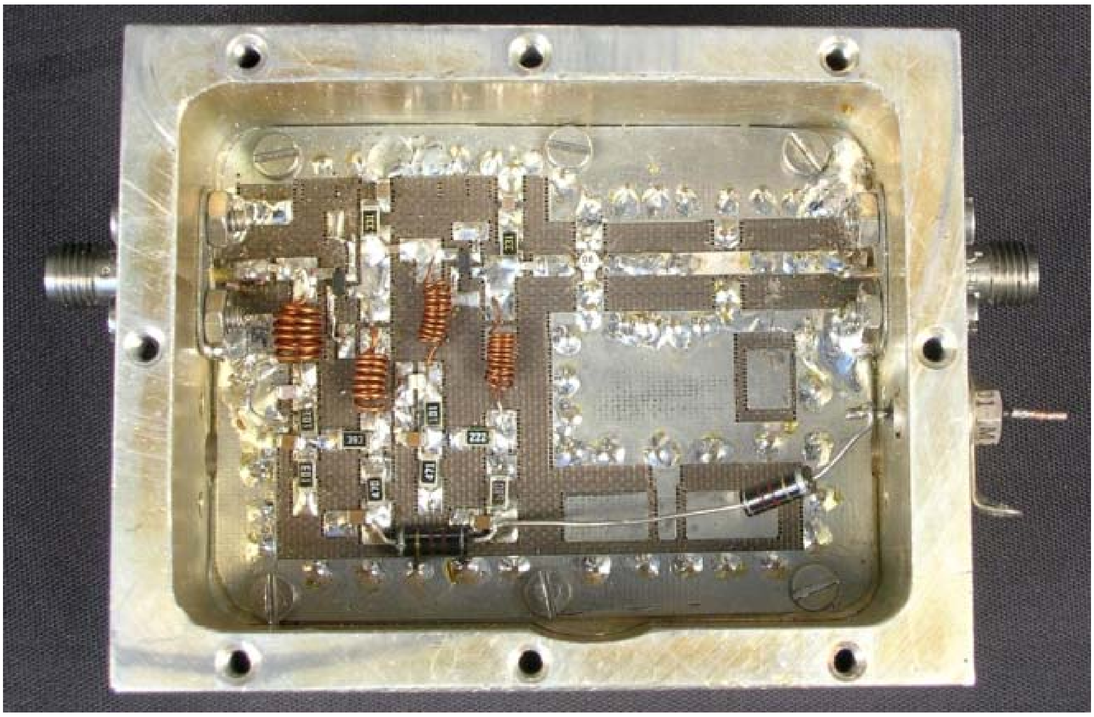


Figure 4.30 - Graphs of LNA obtained with the noise figure Analyzer

4.17 LNA Photos



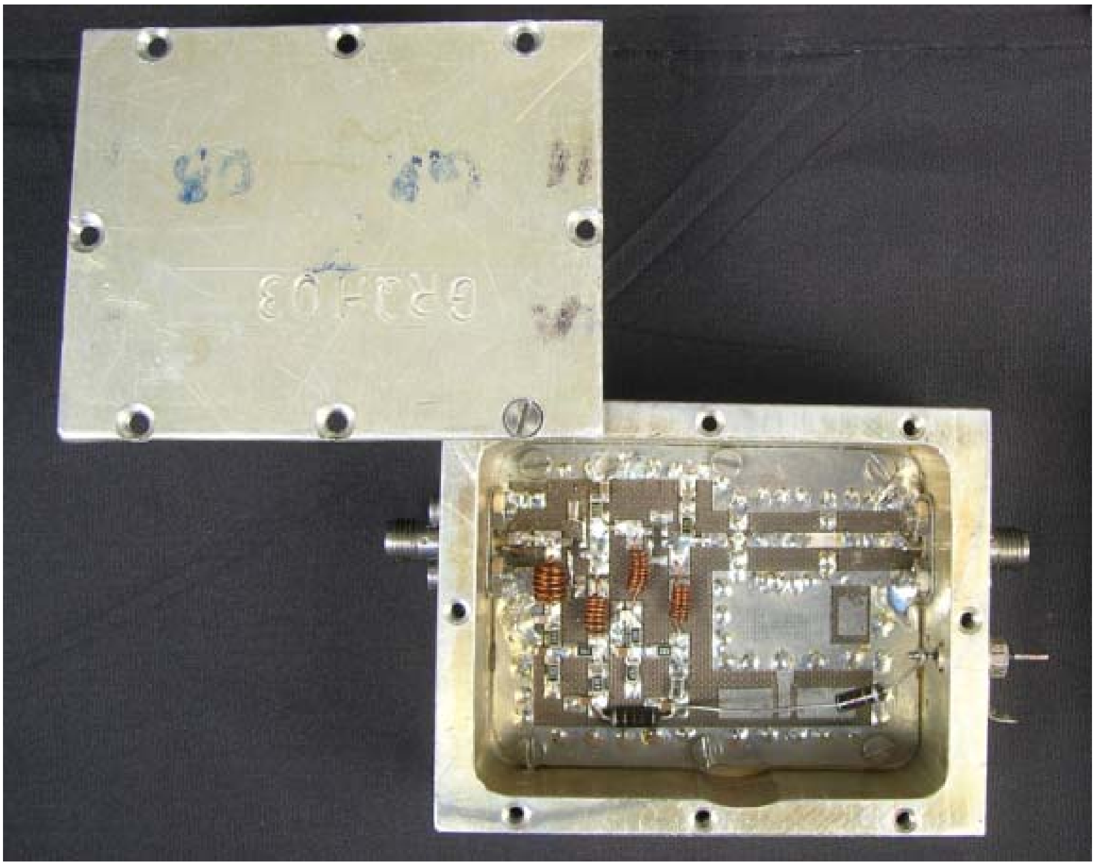


Figure 4.31 - Photographs of LNA – Top view.

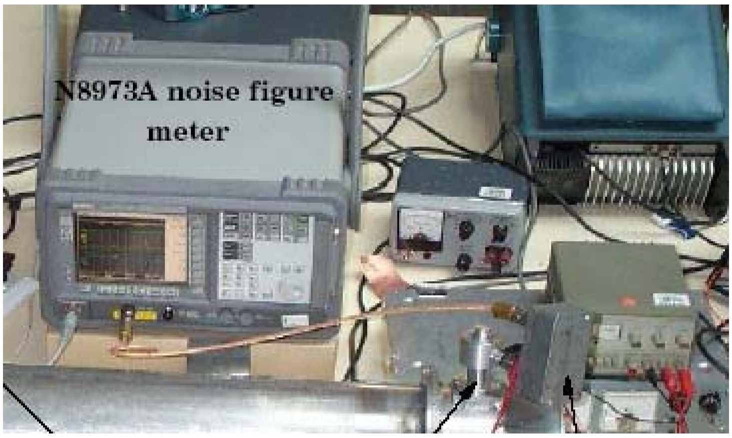
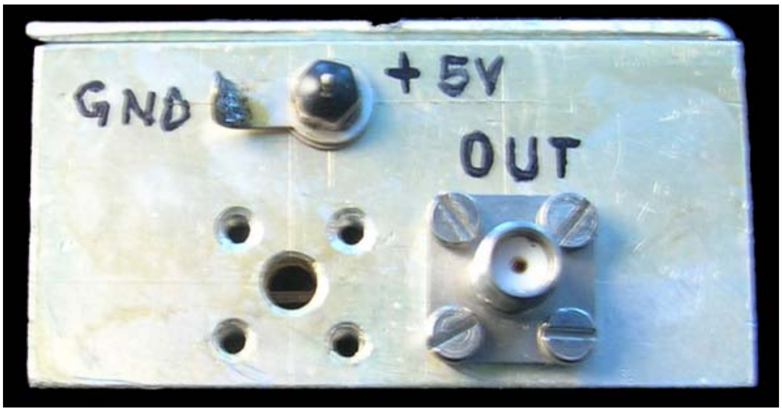
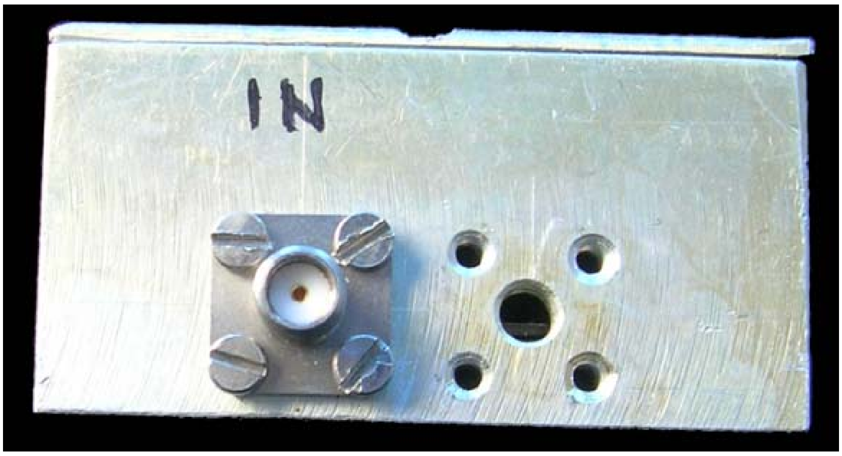


Figure : 4.32 – LNA parameter (Gain, NF, Teffective) measurement by NFA



(a)



(b)

Figure 4.33 – (a) and (b) Photographs of LNA – Front and back view.

CHAPTER 5

TIME MANAGEMENT

Time Management

For the dissertation work total time allotted is of two semester i.e. about a year. The main part of the dissertation are Literature survey and study, Design methodology selection, Device / component / software selection. Design, testing of performance of LNA

Dissertation work	Percentage to total work	Completion Month
Literature Survey	10 %	15 August 2005
Design Concept	20%	10 September 2005
Theoretical & Simulated Design	30%	5 December 2005
Transistor parameter measuring with Network Analyzer.	10%	5 January 2006
Assembly of selected component	10%	10 March 2006
LNA performance testing with Noise meter & adjustments	20%	10 May 2006

Table 4.5 – Time management of the project.

CHAPTER 6

FUTURE SCOPE (SUGGESTED FURTHER IMPROVEMENTS)

6.1 Probe Impedance

We know that the impedance of the probe antenna can be varied by changing the length of the probe. More investigation can be done to determine the ideal probe length for best noise performance.

6.2 Lower Loss Components and Better LNA Design

The newer SM 250 molded inductors that replaced the original Panasonic 15nH higher loss inductors successfully reduced the noise figure. Noise figure may be further reduced by using even lower loss inductors. The Coilcraft 0603HC 15nH inductors that are currently used have ceramic cores, which introduce greater losses than air cores. For instance, Coilcraft's Mini Spring air core 18.5 nH inductors each have a maximum resistance more than 20 times smaller than that of the 0603HC 15 nH ceramic inductors. Air core inductors are larger however and the LNA circuit board would need to be redesigned to allow for larger components. Ultra low loss capacitors such as those provided by ATC and DLI should also be used in RF bypassing and DC blocking.

To simplify troubleshooting, a low loss large RF choke could be placed as close to the input of the transistor as possible while minimizing excess microstrip traces. This would keep the impedance presented to the transistor input as close to the probe impedance as possible. As well, excess inductance could be added to the source leads of the transistor in order to increase stability and power transfer.

6.3 Newer Transistor Technology and Better Selection of Transistors

Inevitably, transistors with better noise performance have been developed since the start of this project. Indium phosphide (InP) HEMTs are an example, with noise figures even lower than that of GaAs HEMTs. However, these devices are still primarily available in the research and development sector of industry, and are more expensive to obtain.

Commercial devices offer a noise figure of approximately 0.5 dB or 35 K noise temperature at 2 GHz. HEMTs made for higher frequencies such as the NE 325 (0.45 dB noise figure or 32 K noise temperature at 12 GHz) and the ATF- 36077 (0.5 dB noise figure or 35 K noise figure at 12 GHz) are possibly worth investigating. I suspect that the transistor used in the 0.3 dB (20 K) commercial LNAs is the ATF-36077 pHEMT, judging from packaging and noise performance. However, there were problems due to the extremely poor input return loss of the device. The datasheet of the NE 325 also specifies poor input return loss as well as low gain at frequencies closer to 1.42 GHz, which could pose design problems. Nevertheless, when selecting a transistor, the theoretical noise figure should be at least 0.2 dB lower than the desired noise figure of the LNA, in order to allow for circuit losses.

CHAPTER 7

CONCLUSION

As part of an effort to double the sensitivity of the Radio Telescope at 500 MHz - 900 MHz , this project aimed to reduce receiver front-end noise temperatures from a mean value of 35 K to 25 K. Currently, the receiver front-ends of the Radio Telescope are LNAs with a mean noise temperature of 35 K and gain of 33 dB. Initially, this project was geared towards LNA design with the Agilent ATF-54143 pHEMT that would have a noise figure of 20 K and a gain of at least 30 dB. There were three main goals:

- Obtaining the correct S-parameters of the Agilent ATF-54143 pHEMT over a wide frequency range
- Calculating the noise parameters of the ATF-54143 at 500 MHz - 900 MHz
- Improving the LNA design for better noise performance & high gain.

Using Thru-Line-Reflect (TRL) calibration and low-loss test fixtures, S parameters of the ATF-54143 were successfully measured from 0.5 to 0.9 GHz. The S-parameters were used to obtain a circuit model of the ATF-34143 transistor. Noise parameters of the ATF-54143 were obtained with Genesys 2005 circuit simulation software using Pospieszalski's method for the intrinsic circuit in the transistor circuit model.

Knowledge of the noise parameters of the ATF-54143 at 500 MHz - 900 MHz GHz permitted design of an LNA with coaxial connectors. The noise figure of the LNA is 0.4 dB or 25 K, a 5 K improvement over the LNAs currently in operation in the feeds of the Radio Telescope. The gain of the LNA is 33 dB, reducing the noise contribution from the next stage to a few hundredths of a Kelvin, which is sufficient for a Radio Telescope front-end. Along with Teresia Ng's proposed antenna design improvements that can lower system noise temperature by 6 K, the LNAs can be incorporated to lower the system noise temperature by 11 K. This would bring the radio telescope system noise temperature at 500 MHz - 900 MHz from 35 K down to 25 K.

The probe LNA design requires further development. Currently, the probe LNA design is a one-stage LNA with the same noise performance as the LNA with coaxial connectors when placed in a waveguide. The gain, however, is only 15 dB because there is only one stage of amplification. There is much potential for even better noise performance than the LNA design with coaxial connectors because of the elimination of sources of loss: fewer components are used at the input of the transistor and there is no coaxial connection between the LNA and the antenna. Suggestions for improvement include adjusting the probe impedance, adding external source inductance, using lower loss components and simplifying the design for troubleshooting purposes by minimizing excess microstrip traces at the input of the transistor. If possible, a transistor with better noise performance (preferably at least 0.2 dB lower than the desired LNA noise figure) and comparable input return loss and gain may be selected. A second stage may be added in order to meet the gain requirement of a Radio Telescope front-end.

CHAPTER 8

BIBLIOGRAPHY (References)

Text books

[1] Microwave Circuit Design – Vendalin & Rhode

[2] Small signal microwave amplifier design

[3] Microwave solid state circuit design

[4] High frequency techniques : An introduction to RF & microwave Engg. – Joseph F. White

[5] Low frequency radio astronomy – Yashwant Gupta.

[6] Low Noise Microwave Transistors & Amplifiers.- Hatsuaki fukui.

[7] Design of LNA for DRAO Synthesis Telescope – Angle Garcia

Magazines / Website

[6] National Research Council of Canada. NRC website about DRAO. URL: http://www.hia-iha.nrc-cnrc.gc.ca/drao/index e.html.

[7] Dominion Radio Astrophysical Observatory. DRAO website. URL: http://www.drao.nrc.ca.

[8] The pseudomorphic HEMT – A.W. Swansun (Magazine : Microwaves & RF, March 1987 page 139 to 150)

[9] Design of Low Noise Amplifier using HEMTs - Satyanarayana (Magazine : RF Design, March 1994 page 98 to 104)

[10] Build An E-pHEMT Low Noise Amplifier – Ian Piple (Magazine : Microwaves & RF, March 2004 page 92 to 102)

[11] Website : http://www.gmrt.ncra.tifr.res.in/

[12] Website : http://www. Drao.ncr.ca

[13] University of Calgary. CGPS public information site. URL: http://www.ras.ucalgary.ca/CGPS.

[14] Arecibo Observatory. Arecibo Observatory website. URL: http://www.naic.edu/.

[15] MIT Haystack Observatory. Undergraduate Research Educational Initiative website. URL: http://web.haystack.mit.edu/urei/index.html.

[16] Annie-Claude Lachapelle, “Note for Low Noise Amplifier,” report, DRAO,

2000.
[17] K. Chang, R. York, P. Hall and T. Itoh, “Active Integrated Antennas,” IEEE Transactions on Microwave Theory and Techniques, vol. 50, pp. 937–944, Mar. 2002.
[18] Marian W. Pospieszalski, “Modeling of Noise Parameters of MESFET’s and MODFET’s and Their Frequency and Temperature Dependence,” IEEE Transactions on Microwave Theory and Techniques, vol. 37, pp. 1340–1350, Sept. 1989.
[19] Agilent Technologies, “Low Noise Pseudomorphic HEMT in a Surface Mount Plastic Package.” ATF-54143 pHEMT Specification Sheet.
[20] Agilent Technologies, “High Intercept Low Noise Amplifiers for 500 MHz through 900 MHz Using the ATF-54143 Low Noise PHEMT.” Application Note AN 1175.
[21] Wolfram Research, “The History of Mathematica.” URL: http://www.wolfram.com/company/history/.
[22] Agilent Technologies, “Fundamentals of RF and Microwave Noise Figure Measurements.” Application Note AN 57-1.
[23] Agilent Technologies, “Low Noise Amplifiers for 900 MHz using the Agilent ATF-54143 Low Noise PHEMT.” Application Note AN 1190.
[24] PaulWade,“Understanding the WB5LUA GaAs FET Bias Circuit.”URL: http://www.w1ghz.org/10g/bias.htm.