Design and Simulation of Hybrid Circuits for LNA used in Wimax Applications

Girijamba C R* and Sathisha R N**
*4th Sem M.Tech, Dept of Telecommunication engineering, Siddaganga Institute of Technology, Tumakuru
girijambacr@gmail.com
** Assistant Professor
Dept of Telecommunication engineering, Siddaganga Institute of Technology, Tumakuru
satisharyan@gmail.com

Abstract: The objective of the project is to design a Low Noise Amplifier(LNA). The amplifier is designed for different specific parameters depending on the application. Typical parameters are maximum transducer gain, output power, low noise, circuit stability. Here a LNA is designed to obtain an optimum gain with minimum noise figure for a design frequency. In the project transistor BFP 420 with a DC operating point (VDS=1.5v, ID=15ma) and designed for a maximum center frequency(f=2.4Ghz). The entire design is carried out with the AWR Microwave Office design tool and the simulation results for each of the stages are clearly presented.[1].

Keywords: low noise amplifier; WiMAX; AWR MWO; Gain; noise figure.

Introduction

The demand for compact, high gain, low noise and low cost amplifier has increased significantly for the antenna design with the improvement in the wireless communication technology in recent years. Here we described the bilateral design procedure of a low noise amplifier for given parameters. Maximum transducer gain and less noise figure should be achieved. Low Noise Amplifiers (LNA) are used for amplification of extremely weak input signals. The LNA is usually placed in the first stage of a microwave receiver circuit because the first stage decides the quality factor of the receiver. The first stage affects the overall noise figure of the receiver and thereby placement of a LNA in this stage reduces the overall cost of the amplifier. The most important parameters and keywords for a LNA design are gain, noise figure, non-linearity, stability and matching. The applications of LNA are in GPS receivers, Cellular phones, Wireless LANs, satellite communications etc. **WiMAX** is an acronym for worldwide interoperability for microwave access. The system is based on the **IEEE 802.16** standard that is also known as Wireless **MAN**.[3] WiMAX works similar to wifi protocol as it can be used in wireless networking. The second generation protocol i.e. WiMAX it effectively utilizes bandwidth, avoids interference and it allows higher data rates over long distances. WiMAX channel capacity initially offer about 40 Mbps capacity per wireless channels for both fixed and portable applications. The properties of WiMAX are it has broad bandwidth and a data rate of 134Mbps in 28MHz channel and it supports multiple services such as IPV6, ATM, Ethernet etc. WiMAX supports adaptive antennas and space time coding. The advantages of WiMAX are last mile connectivity, routing between the networks, flexibility and scalabilty. WiMAX can support voice and video as well as Internet data. WiMAX can operate at a variety of spectrum bands: 2.3 GHz, 2.5 GHz, and 3.5 GHz for licensed band and 5.8 GHz for unlicensed band. LNA is a key component in WiMAX systems, which is placed at the front-end of a radio receiver circuit, so that losses in the feed line become less critical. When using a low noise amplifier, noise is reduced by the noise figure and the gain of the amplifier. As the first stage of the receiver, LNA's are required to have a high gain and low Noise Figure.[1-3].

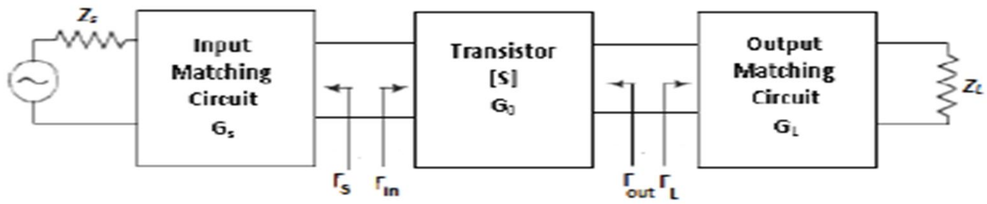

Fig 1: General transistor amplifier circuit

LNA design flow chart

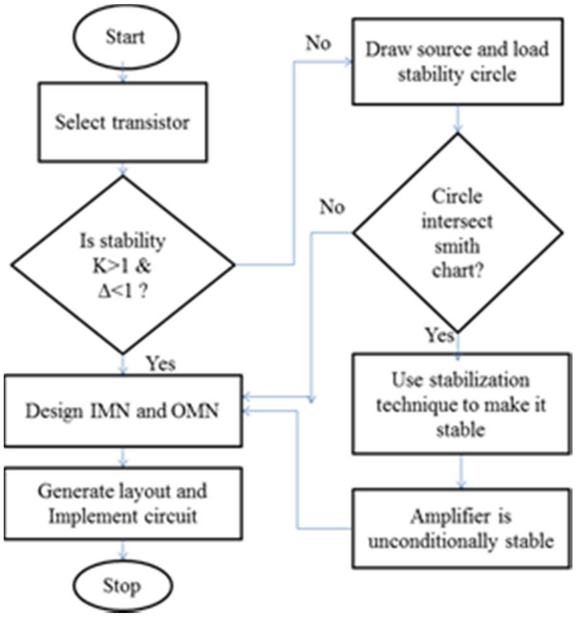

Fig 2: flowchart for LNA design

LNA design steps

- The first step in designing a LNA is choosing a suitable transistor.[2]
- After choosing a transistor first check for the stability, the stability factor K>1 and δ<1 for unconditional stability
- If the stability condition didn't satisfy, then draw load and source stability circle else draw input and output matching circuit.
- If the circle intersect the smith chart implies that transistor is unstable. So use the stabilization technique to make it stable.
- Finally implement the circuit and simulate.

Design Using S-Parameters

S-parameters are widely used than with Y-parameters because of their ease to measure and work. They are easy to comprehend, convenient and provide a wealth of information at a glance. [4]

S-parameters can be characterized by their two port networks. S-parameters help in calculation of maximum available gain, potential instabilities, input and output impedances and transducer gain. S-parameters also allow the calculation of optimum source and load impedances, to choose the source and load impedances for a specified transducer gain.

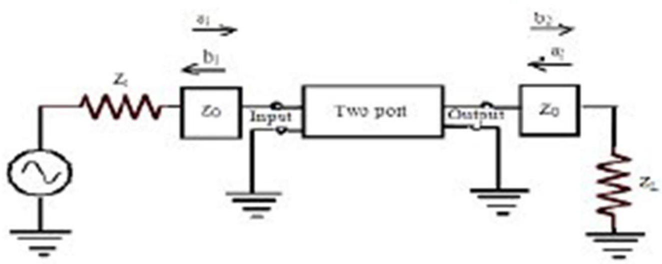

Fig 3: Two port network

Given specifications are: Transistor BFP-420, Frequency=2.4GHZ

Bias: VDS=1.5v, ID=15mA

The S-parameters of the Transistor for the given specifications are shown in table I:

Table I: S-parameters

S-Parameter	Polar Form	Rectangular Form
S_{11}	$0.494\angle 160.90$	-0.467+0.1617i
S_{12}	$0.075\angle 42.32$	0.056+0.0508i
S_{21}	$5.29\angle 52.83$	2.66+4.576i
S_{22}	$0.186\angle 227.91$	-0.125-0.1384i

1. Reflection co-efficients are found as:

$$\Gamma_s = \frac{-B1 \pm \sqrt{B_1^2 - 4|C_1|}}{2|C_1|} = -0.7160 - 0.1376i = 0.729\angle -169.2$$

$$\Gamma_L = \frac{-B_2 \pm \sqrt{B_2^2 - 4|c_2|}}{2|C_2|} = 0.0147 + 0.5727i = 0.577\angle 88.52$$

Where,

$$B_1 = 1 + |S_{11}|^2 - |S_{22}|^2 - |\Delta|^2 = 1.0621$$

$$C_1 = S_{11} - \Delta S_{22}^* = -0.4945 + 0.0956\, i$$

$$C_2 = S_{22} - \Delta S_{11}^* = 0.0078 - 0.2739\, i$$

$$\Delta = (S_{11} * S_{22}) - (S_{12} * S_{21}) = 0.116 - 0.350\, i$$

2. To find the length of the stub lines:

For Source:

$$\Gamma_S = -0.7160 - 0.1376i$$

$$T_{l_{ocl1}} = 0.183$$

$$El = 0.183 * 360$$

$$El = 65.88^{\bullet}$$

$$T_{l_{inl2}} = 0.04$$

$$El = 0.04 * 360$$

$$El = 14.7^{\bullet}$$

For Load:

$$\Gamma_L = 0.0147 + 0.5727i$$

$$T_{locl1} = 0.344$$

$$El = 0.344 * 360$$

$$El = 125.5^{\bullet}$$

$$T_{linl2} = 0.05$$

$$El = 0.05 * 360$$

$$El = 18^{\bullet}$$

Theoritical calculation of Gain

$$G_S = \frac{1}{1 - |\Gamma_S|^2} = 1.29\, dB$$

$$G_0 = |S_{21}|^2 = 14.33\, dB$$

$$G_l = \frac{1 - |\Gamma_l|^2}{|1 - S_{22}\Gamma_l|^2} = 1.21\, dB$$

$$GT_{\max} = G_s * G_0 * G_l = 16.7 dB$$

Matching network

Network matching is a matching of source impedance to a load impedance to maximize the power transfer between source and load. It can be done in two ways either by inserting capacitive and inductive elements or by inserting a combination of transmission line and a stub line that produces same impedance effect as of the reactive elements and matches the network to the characteristic impedance. Using stub line matching technique the lengths of the stub lines can be found using the smith chart. [2]

The input and output matching networks along with the specified transistor BFP 420 along with DC operating point is shown in Fig 4:

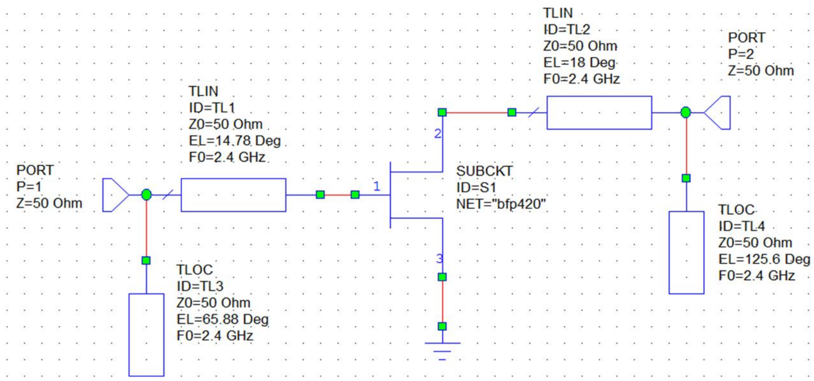

Fig 4: Schematic of Ideal Low Noise Amplifie

The ideal response of amplifier circuit is shown in Fig5.

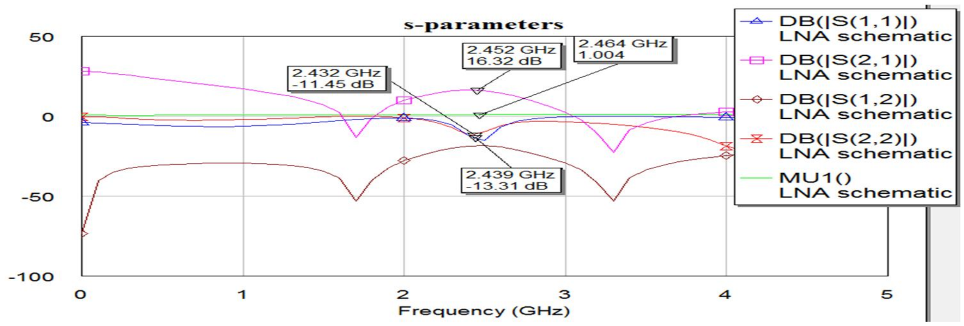

Fig 5: Response of Ideal Amplifier

From the above Fig 5 the calculated theoretical gain 16.7dB is verified with the simulation result.

Stability

Stability is the ability of an amplifier to maintain effectiveness in its nominal operating characteristics inspite of large changes in the environment such as physical temperature, signal frequency, source or load conditions etc.[7]

Stability refers to an amplifier's immunity to causing spurious oscillation. The first step in designing a LNA is determining its stability factor. It is important to check the stability of an amplifier because otherwise the amplifier may function like an oscillator. [4] The stability of the amplifier increases for an increase in the frequency. At frequency of 1GHz amplifier has stability lower than one and increases until the maximum is reached at our desired frequency of 2.4GHZ.

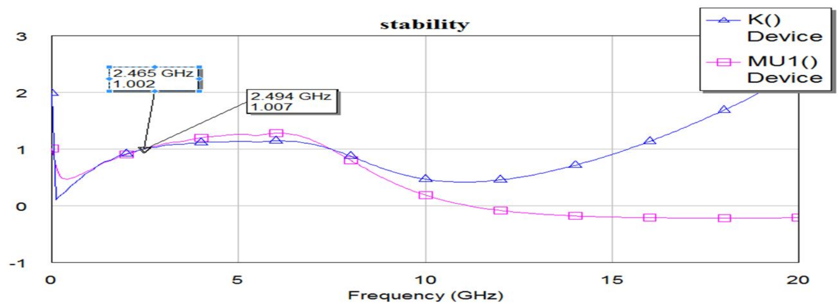

Fig 6: Stability k and μ sweep

The circuit is unconditionally stable as the values of k and μ are greater than one at the designed frequency. The stability decreases down to one at higher frequencies.

In order to obtain stability also for low frequencies such as at 1 GHz the following circuit was introduced using a resistor and a capacitor in parallel. The best stability was obtained by using the tuning tool of MWO.

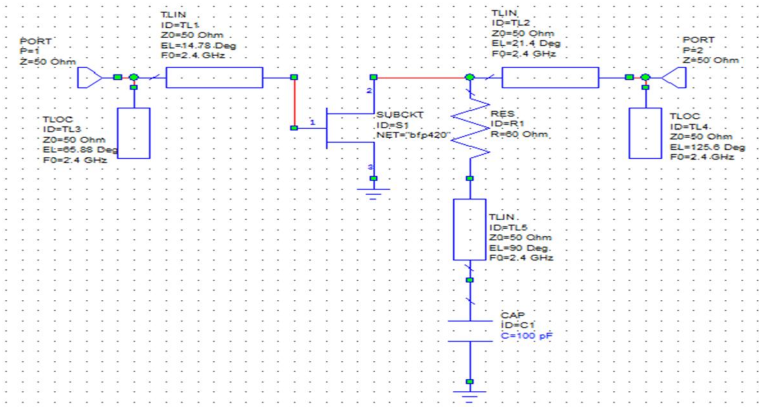

Fig 7: Stability circuit with improved LNA

The s-parameters and the resulting stability for the above circuit is shown in Fig 8.

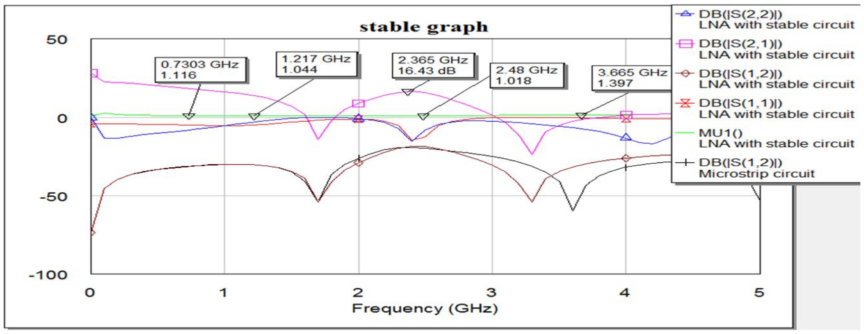

Fig 8: S-parameters and µ-sweep for improved LNA

From the Fig 8 it is seen that the circuit is stable at all the frequencies.

Design using stability network and microstrip lines

By using real transmission lines i.e. Microstrip lines the design can be improved. We use the substrate GaAs here with dielectric constant=3.55, tan δ=0.0025 and copper thickness=17µm.

µ-test for unconditional stability

The theoretical stability can be verified by using µ-test. The value of µ should be greater than one in order to prove that the circuit is unconditionally stable.[8]

$$\mu = \frac{1 - |s_{11}|^2}{|s_{22} - \Delta s_{11}^*| + |s_{12} s_{21}|} = 1.74$$

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

Microstrip lines design circuit

The design of low noise amplifier using microstrip lines is shown in Fig 9.

Design Procedure

- A resistor and capacitor is added in parallel at the output port in order to have the stability also at low frequencies.
- The ideal transmission line is added in series with the parallel resistance because the gain decreases when resistor is added.
- The ideal transmission line is replaced with the microstrip line(MSTUB) which can be understood from fig 9.The dimension of this line can be calculated using Txline calculator in AWR. In the below fig10 shown the general physical characteristics were applied to all Microstrip lines.
- To add resistor and other additional components in the circuit extra Microstrip lines are added in the circuit.

- In the fig 9 we can see that all the lines were connected with the help of T-junction.
- When a T-junction is inserted the extra length is added. In order to compensate that extra length a short open line is added at the 3rd port of T-junction.This can be done by using tuning tool of MWO.[5]

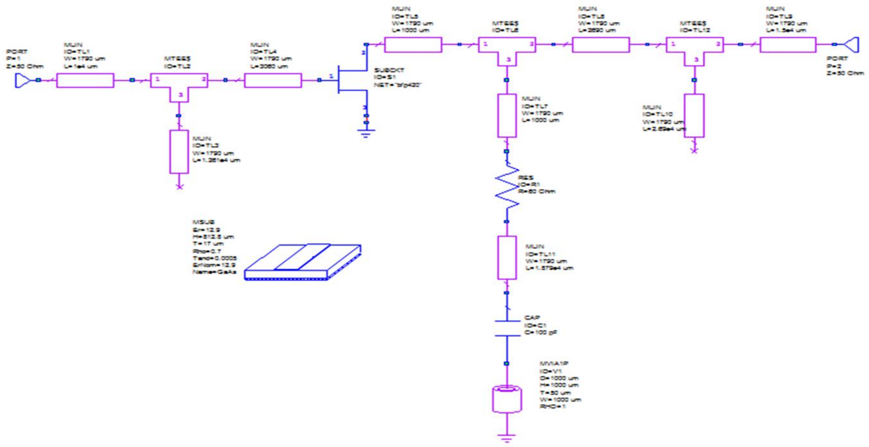

Fig 9: Design of LNA using Microstrip Lines

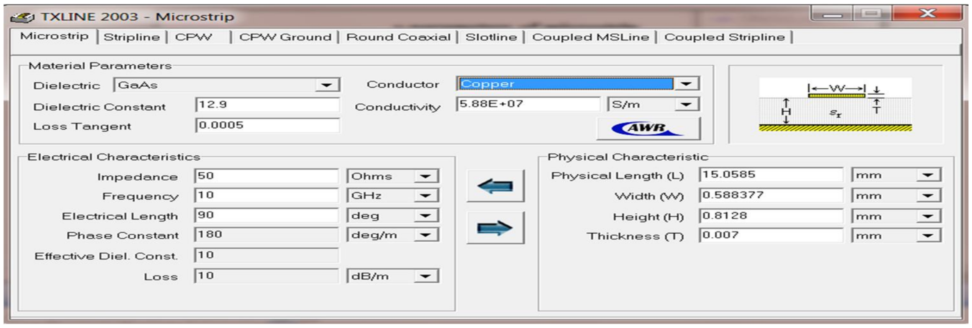

Fig 10: Txline calculator

The stability µ-sweep, noise figure, s-parameters are simulated for the above amplifier circuit with microstrip lines and can be observed in the Fig 11.

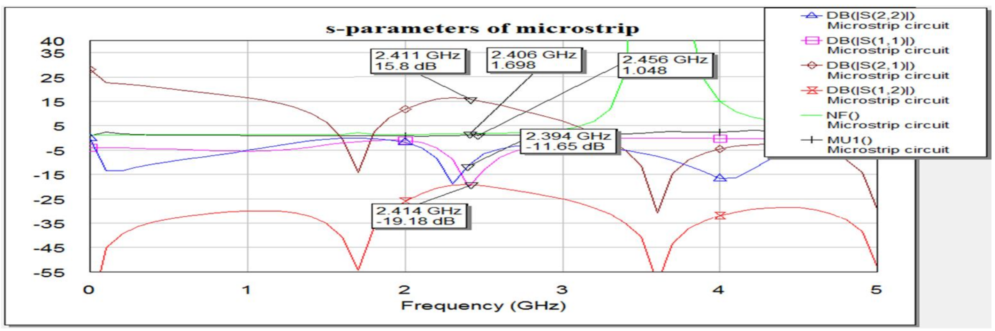

Fig 11: Simulation results of Microstrip circuit

Design of amplifier using biasing network

A radial stub line of length λ/4 is added at both the ports which acts as an open line for the RF signal. An additional line is connected to provide a port to connect the DC power source. Blocking capacitors are added to isolate the RF signal from the DC power source at the base and collector terminal.

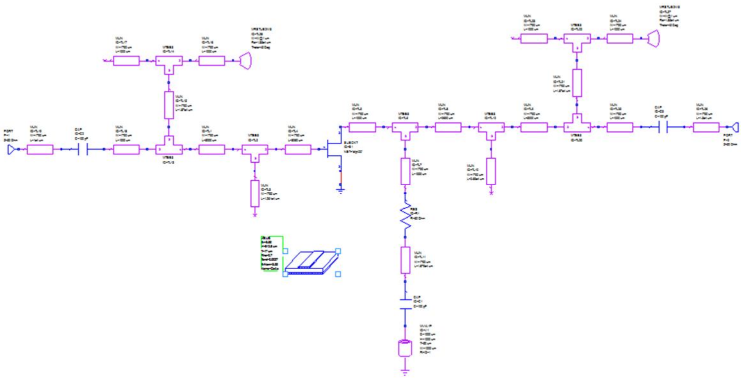

Fig 12: Design of LNA using Biasing Network

The stability μ-sweep, S-parameters and noise figure for the above biasing network is shown in Fig 13.

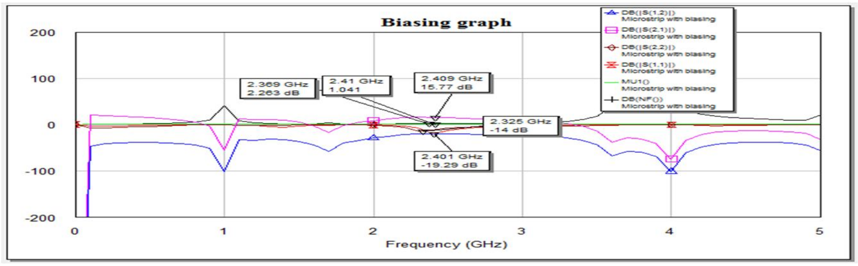

Fig 13: S-parameter sweep for the biasing network

Cascading of two stage amplifier circuit

In a single stage amplifier we cannot get sufficient gain and bandwidth and moreover we didn't have matched input and output impedance. For better performance and amplification multiple amplifiers are cascaded to overcome such problems. To increase the voltage gain of the amplifier multiple amplifiers are cascaded together.

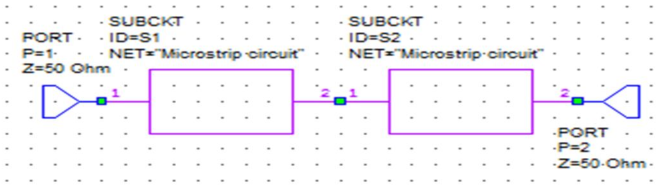

Fig 14: cascading of two stage amplifier circuit

The S-parameters and noise figure for the above circuit is shown in Fig 15.

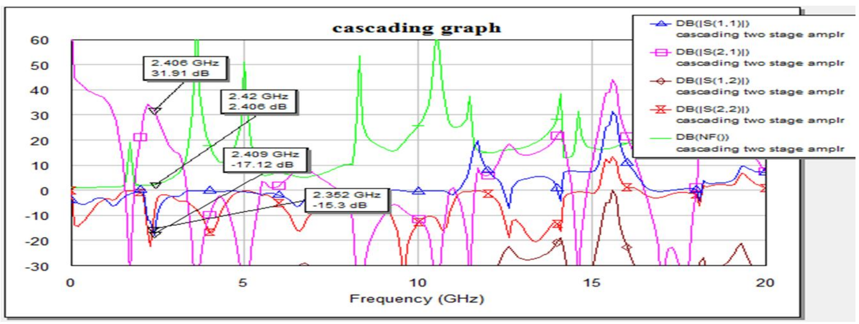

Fig 15: s-parameters of cascaded amplifier

From the above figure it is seen that the gain of the amplifier is increased and a quite increase in the noise figure.

Recent work

In paper [2] the LNA has been designed for low frequency of 1800MHz and CMOS 0.35μm technology has been used and for this low frequency the noise figure achieved is high. In [2] wide band LNA has been designed using HEMT transistor and coupler for low frequency band only. Here the coupler is used to achieve high gain but it is not cost efficient and complexity is also more. In [8], the author designed LNA using MESFET transistor for 5 to 6 GHz frequency range. For matching network LC combination has been used. The Gain achieved is 15.83db. The author achieved the poor stability in the design and noise figure increases as the frequency is increased. In [7], different combination of matching networks has been used for the design of LNA. The gain achieved is minimum and noise figure is high at 6 GHz frequency. L-L Duality matching network has been used for input and output matching [4]. The stability achieved is minimum and gain also minimum at 6GHz frequency. The LNA proposed in [1] does not work in low frequency and consumes high power.

Conclusion

The objective to design a low noise amplifier for maximum gain was reached by using computation methods from Microwave office as well as the Smith chart. With the help of the theoretical design the ideal transmission lines were realized. The gain was optimized by introducing resistors at the output port. By using the Microwave Office tuning tool the stability was improved for operation at lower frequencies. The microstrips lines were successfully implemented from the ideal lines, further real microstrip elements were added and optimized for stability using the MWO tuning tool. The biasing network for RF-DC decoupling was included from which the final layout of the low noise amplifier obtained. Finally a gain of 16.5dB and noise figure around 2.2 dB and good return loss is obtained.

Future work

In the future, the designs will be fabricated and measured. The fabricated results will be compared to the simulation results for analysis. The fabrication results are important for this project commercialized value for future used.

Acknowledgment

I express my deep sense of gratitude to Dr. Sree Sree Shivakumara Swamigalu, president, Siddaganga Esducation society, for his blessings and for providing me with an opportunity to fulfill my most cherishable desires of reaching my goal.
I would like to express my sincere gratitude to Prof. Sathisha R.N. for his timely guidance, valuable suggestions, advice and encouragement during the course of my project work.

References

[1] Ishaan Biswas, AnupDeka, SC Bose, “Design of a 2.3 GHz Low Noise Amplifier for WIMAX application”, international conference on device, circuits and systems,2012, March 15-16 , pp. 105-109.
[2] Karpagam S, Sampath P, “Design and implementation of Low Noise Amplifier for 1800MHz Band GSM receiver”, IJAVD vol. 4, No. 2, 2014, pp. 95-98.
[3] Madhura P Joshi, Savita B Bhosale, “ Efficient designing techniques for Low noise Amplifier”, Proceedings of 22nd IRF International conference, Pune, 2015, pp. 106-108.
[4] S.Vimal , Dr.M.Maheshwari, “Design and performance improvement of a low noise amplifier with different matching techniques and stability network”, IJOER Vol-2, Issue-3 March- 2016.
[5] Guillermi Gonzalez, “Microwave transistor amplifiers analysis and design” second edition, Prentice Hall, 1997.
[6] Mike Golio, Janet Golio“RF and microwave passive and active technologies” second edition, ISBN 9780849372209, December 22, 2007 by CRC Press
[7] Y.H. Yu, Y.S. Yang, Y.J.E. Chen,” A compact wideband CMOS low noise amplifier with gain flatness enhancement”, IEEE. Journal of solid state circuits, Vol. 45, no. 3, 2010, pp-502-509.
[8] Neha Rani, Suraj Sharma, “ Design of Low noise Amplifier at 3-10 GHz for Ultra wide band Reciever”, International Journal of Innovative Research in Computer and Communication Engineering, Vol 1, Issue 7, 2013, pp. 1401-1409.
[9] Niti Mohan, Vaithianathan.V “Noise Analysis of the Input Matching Circuits for UWB Low Noise Amplifiers” at International conference on Communication and Signal Processing, April 3-5, 2013,India , 978-1-4673-4866-9/13/$31.00 ©2013 IEEE Pg :545-550