International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-11 Issue-3, January 2022

Design of 2.4 GHz LNA of 400 MHz Bandwidth

Abhay Chopde, Prashik Sadar, Ashutosh Sabale, Piyush Thite, Raghvendra Zarkar

Abstract*: Low Noise Amplifier (LNA) is the most important front-end block of the receiver. LNA's Noise figure (NF) and Scattering Parameters affect the overall performance of the whole receiver circuit. Nowadays in the era of 5G technology, The quality of data that is being transmitted is increased. So there is a need for higher bandwidth to transfer data with higher speed. In such a case, communication blocks need an update. The research is carried out for the advancement of the LNA. The primary goal of LNA design is to lower the Noise Figure and return losses. The paper aims to design a 2.4 GHz LNA having a bandwidth of 400 MHz. The circuit is designed with the help of single-stub microstrip lines. We tried to keep the length of microstrip lines as minimum as possible. The transistor ATF-21170 Gallium Arsenide Field Effect Transistor (GaAs FET) is used in this work. The circuit is simulated in the Keysight Advance Design System (ADS). The amplifier is manually designed using standard methods. LNA is unconditionally stable for the frequency range of 2.2 GHz to 2.6 GHz. To build impedance matching circuits of the amplifier smith chart is used. It is observed that the LNA gain (S21) is greater than 15.3 dB, NF less than 1.2 dB, Input return loss (S11) is less than -13.3 dB, Output return loss (S22) is less than -17.1 dB over the 400 MHz bandwidth ranging from 2.2 to 2.6 GHz. This has, to the best of the authors' knowledge, not been presented in literature before.

Keywords*: Cascode, ATF-21170, Frii's Formula, Miller Effect, Scattering (S) Parameter.

I. INTRODUCTION

Low Noise Amplifier (LNA) is the very first block of the receiver in wireless communication. Wireless products such as cellular phones, global systems for mobile communication, global positioning satellite, wireless local area network, etc. became an everyday part of people's lives, along with the size of the wireless device, there is a need for higher performance at low power consumption and cost. Nowadays the systems are moving towards automation and concepts of cognitive learning (cognitive radar and cognitive satellite communication), it becomes more important to maximize the improvement of performance parameters of the device. LNA is one of the most important blocks in RF circuit design because the receiver's sensitivity depends on the noise figure and power gain of LNA. The Noise performance of the receiver is very much more influenced by the noise fig of LNA. It also contributes to the linearity of the entire system. There are diverse topologies to design LNA and a lot of swapping among the values of the parameters. A circuit with minimum noise and better performance is the main motive. LNA with better performance is needed because it is the first stage amplifier in the receiver. According to Frii's formula, the first stage of the receiver has more contribution to the overall noise figure. That's why it is important to design LNA with a minimum noise figure. There are four main parameters of the LNA that are gain, Noise figure, return losses, and stability. These parameters are defined by S parameters, S21 is gain, S11 is return loss of input, S22 is return loss of output. For stability, we have Rollet's stability factor, and it should be greater than 1. A design of common source LNA with low NF and high Gain is presented in this paper. Simulations of the proposed LNA design are accomplished in ADS software.

Manuscript received on January 01, 2022.
Revised Manuscript received on February 05, 2022.
Manuscript published on February 28, 2022.
* Correspondence Author
Abhay Chopde*, Department of E & TC Engineering, Vishwakarma Institute of Technology, Pune, India. Email: abhay.chopde@vit.edu
Prashik Sadar, Department of E & TC Engineering, Vishwakarma Institute of Technology, Pune, India. Email: prashiksadar@gmail.com
Ashutosh Sabale, Department of E & TC Engineering, Vishwakarma Institute of Technology, Pune, India. Email: ashutoshsabale19@gmail.com
Piyush Thite, Department of E & TC Engineering, Vishwakarma Institute of Technology, Pune, India. Email: piyush.thite18@vit.edu
Raghvendra Zarkar, Department of E & TC Engineering, Vishwakarma Institute of Technology, Pune, India. Email: zraghvendra121@gmail.com

II. LITERATURE REVIEW

In this paper, DC biasing, stability bias and matching of input and output using microstrip lines are the steps that are used to analyze and design LNA. As LNA is present in the front stage of the receiver, we required a noise figure as minimum as possible, for calculation of the noise figure, we use Frii's formula [1]. The author in [2] implemented the design by using GaAs. He talks about various methods to improve gain and noise figure. LNA is implemented using the low power differential method in the technology of 65nm CMOS in [3]. It does not consist of inductors. The structure is cross-coupled. The output stage of this structure is capable to drive current in either direction for the given load. It is required to improve transconductance and partial noise-canceling considering power consumption is kept low. A cascode amplifier is used to minimize the Miller-effect, also to switch current between two sides of the differential pair with a current source. It helps to enhance BW and gain. This technique provides better results and signifies the overall performance adjustment after we optimize the biasing under the shortage of power. The required factors to achieve Unconditional Stability along with imperishable gain and low noise figures has explained in [4]. Guillermo Gonzalez talked about the conversion of the ideal transmission line into practical microstrip lines for the implementation of LNA [5]. The design of CMOS LNA based on 130nm CMOS technology has presented in [6]. This LNA has an inductive degeneration topology which gives enhanced gain and noise figure. Simulations are performed with the help of the ADS. The design of LNA exhibits a gain of 20 dB and noise figure nearly 2 dB. The LNA design in this paper is implemented for Bluetooth applications In research [7], The differential LNA is designed.

It is based on 180-nanometer technology. This is the narrowband design with a center frequency of 2.4 GHz. It has inductive degeneration Topology. The proposed design is stable at 2.4 GHz frequency as the value of Rollet's factor is greater than one. This design gives decent values of parameters. S11 is less than -14 dB, S22 is less than -10 dB. It has gain greater than 12 dB and noise figure nearly 1.9 dB. The design and simulation are accomplished with the help of the Cadence Virtuoso IC tool. The circuit needs a power supply of 1.8 Volt. Feedback containing parallel resistance is used in [8] to design LNA as most of the LNA's are developed on series inductive feedback topology. It enhances the noise figure and gain. But using this topology it is difficult to get the value of the inductor for source degeneration which is important for matching the impedance. Therefore, we can design inductors that are inside of Integrated Circuits. This causes the degrading of the NF in simulation. Therefore, the author uses parallel resistive feedback topology. A noise figure of 4.20 dB and a gain of 12.40 dB are seen in the simulation results.

LNA which has a 60 GHz center frequency has its power consumption is low and is designed for wireless communication [9]. 65nm CMOS technology is used for this design. There are two stages in this LNA, an input stage and gain stage with the first one having capacitive cross-coupling technique and the second one which makes the use of current-reuse techniques. When this circuit is executed, it provides a gain of 15dB and a noise figure of 4.7dB.

Simulation of various devices is explained in [10] using ADS software.

III. CIRCUIT DESIGN AND SIMULATION

A. Device Selection

The selection of the transistor plays a major role design of LNA. At a certain frequency, every transistor has its maximum available gain and minimum noise figure. We cannot have a gain greater than max available gain and NF less than minimum NF. We select the ATF-21170 transistor for the design as its specifications were matching as per our required parameters. One factor is also that the minimum noise figure of the transistor should be half that of our required noise figure.

Specifications of the ATF-21107

Frequency Range of Transistor: 0.5 GHz to 6GHz.
Low noise figure: 0.9dB at 4GHz
Gain: 13 dB at 4Ghz [13]

The ATF-21170 is a high-performance GaAs Schottky barrier gate FET. The gate length of the GaAs FET component is three hundred nanometers and an overall gate periphery of seven hundred and fifty micrometers[13].

B. Biasing

Biasing of Transistor is done to keep the proper flow of direct current (DC) and to preserve proper collector to emitter voltage when we apply the signal. This is the definition of a bias circuit. The Biasing circuit of LNA was implemented in the ADS workplace. The lumped components were used for this circuit which is present in the component list in ADS

we first implemented it by using lumped networks as shown in the figure. 1

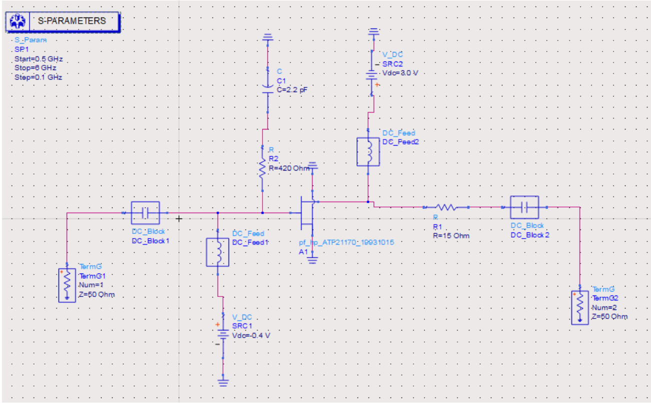

Fig. 1 Biasing circuit

The circuit then changed by replacing the lumped components with distributive components as shown in figure 2.

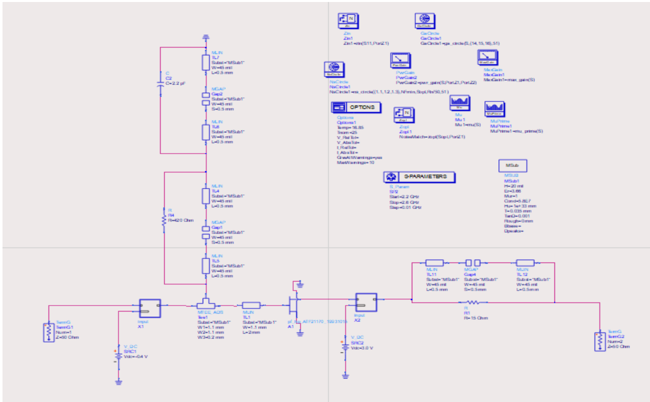

Fig. 2 Stable circuit with distributive components

C. Stability Consideration of Low Noise Amplifier

Stability analysis plays a very important role in the design of LNA as well as any other RF circuits. For stability, we need S parameters values.To check the stability of the circuit we have two stability factors μ and Δ. Stability and matching conditions are the main objective of any LNA along with Noise Figure. If the circuit is unconditionally stable it means that the circuit should be stable for the entire required frequency range and even if we change the load, the circuit should remain stable. For linear two-port devices, to check the stability, we have new criteria other than Rollet's stability factor. In the paper [11], the author proposes the single parameter μ (mu) which is the function of the S parameter and is defined as.

$$\mu = \frac{1 - |S_{11}|^2}{|S_{22} - \Delta S_{11}| + |S_{12}S_{21}|} \quad (1)$$

Where S_{11}, S_{12}, S_{22}, S_{21} are the scattering parameters and $|\Delta| = |S_{11}S_{22} - S_{12}\, S_{21}|$. $|\Delta|$ should less than 1

So if μ>1 then we can say that the circuit is unconditionally stable. We can say that Stability is greater if μ is greater. In this study, two stability variables are considered: μ for the load and μ' for the source. In our simulation, as shown in figure 3, we can see that both stability factors are greater than 1 from 2.2 GHz to 2.6 GHz. So, we can conclude that the circuit is unconditionally stable.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

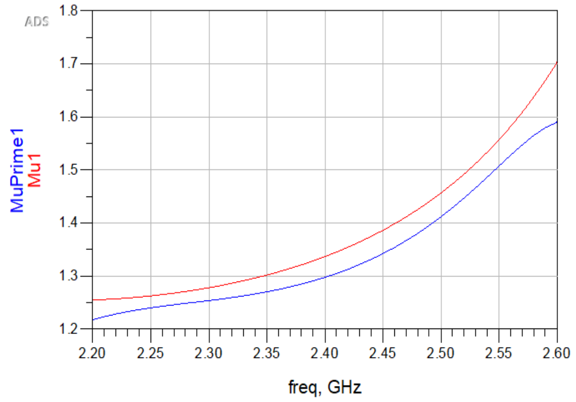

Fig. 3 Stability Factor Plot

D. Input-Output Matching Network

In our design, we took the standard value of source and load that is 50 ohms. To avoid reflections, we need to match the impedance in the input as well as in the output side. After matching impedances, we will get a minimum noise figure and maximum power transfer. and for this purpose, with the help of the Advance Design System, we analyze noise and gain circles. The Centre and Radius of both circles are given by [12].

$$C_{gs} = \frac{g_s|S_{11}|}{1-|S_{11}|^2(1-g_s)} \qquad (2)$$

$$r_{gs} = \frac{(1-|S_{11}|^2)\sqrt{1-g_s}}{1-|S_{11}|^2(1-g_s)} \qquad (3)$$

$$C_F = \frac{\Gamma\text{out}}{1+N_i} \qquad (4)$$

$$R_F = \frac{\sqrt{N_i^2 + N_i(1-|\Gamma\text{out}|^2)}}{1+N_i} \qquad (5)$$

- Here constant gain circle's center is represented by Cgs.
- Constant gain circle's radius is represented by Rgs.
- Constant noise circle's center is represented by C_F.
- Constant noise circle's radius is represented by R_F.

Figure 4 shows the noise and gain circles. The value of gain is the same throughout the circumference of the circle having a certain value of radius. Same for the noise circle. An increase in the radius of the gain circle will decrease the gain. An increase in the radius of the noise circle will increase the noise. The requirement is more gain and less noise figure. So we need to compensate between noise and gain. We need to choose a point where we will get decent values of both parameters. After selecting the point, we can calculate input reflection coefficient (Γs) at that point. With the help of Γs, we can calculate the output reflection coefficient Γout with the help of the following relation.

$$\Gamma\text{out} = S_{22} + \frac{S_{12}S_{21}\Gamma s}{1-S_{11}\Gamma s} \qquad (6)$$

So after balancing noise and gain and analyzing the circles we got the values for input matching and output matching
Input matching = 15-5*j
Output matching = 49+24*j

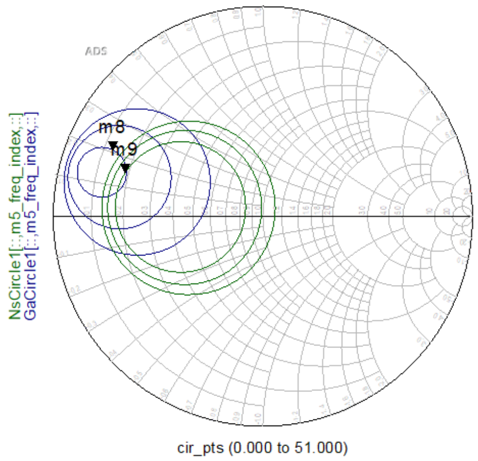

Fig. 4 Noise and Gain Circles

With the help of the smith chart utility provides by ADS, we implemented both matching networks using distributive components such as open stub and short stub.
The transmission lines in the matching circuit were impractical and they were converted into practical microstrip lines with the help of LineCalc provided by ADS which helps to find out width and length by giving electrical specifications as input. Input Match Circuit

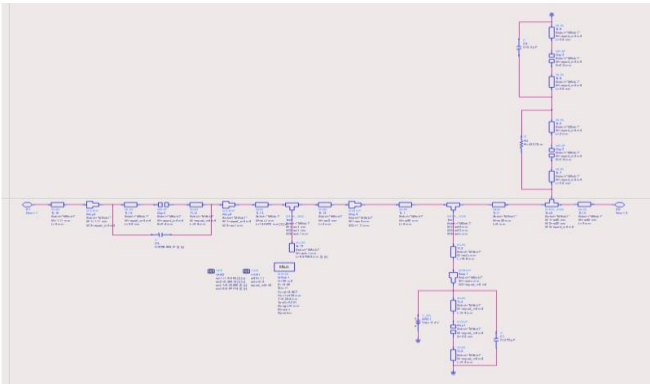

Fig. 5 Input Match

Output Match Circuit

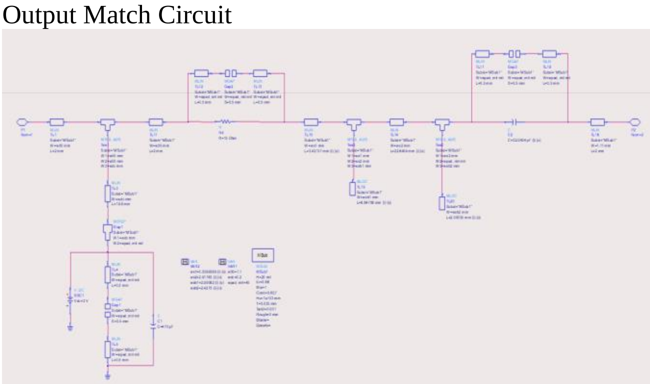

Fig. 6 Output Match

Input-Output Return Loss Figure 7 depicts the simulated consequence of this design's input-output return losses. Over the range of 400 MHz bandwidth, S11 is less than -13.3 dB and S22 is less than -17 dB

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

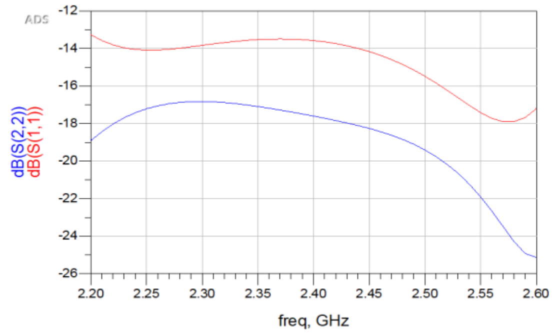

Fig. 7 S11 and S22 Plots

E. Noise Figure

To calculate the noise figure of an amplifier, we need to divide the SNR of input by SNR of output [5].

$$NF(dB) = 10\log_{10}\frac{SNR}{SNR_{out}} \qquad (7)$$

Here SNR is signal to noise ratio.

For amplifier having two stages, the formula which will help to calculate noise figure is given by

$$NF = F_1 + \frac{F_2 - 1}{G_{A1}} + \frac{F_3 - 1}{G_{A1}G_{A2}} \qquad (8)$$

Here F_i and G_{Ai} are the noise factor and available power gain of the i^{th} stage in multistage amplifier.

Figure 8 shows the simulation output for NF over the frequency range 2.2 to 2.4

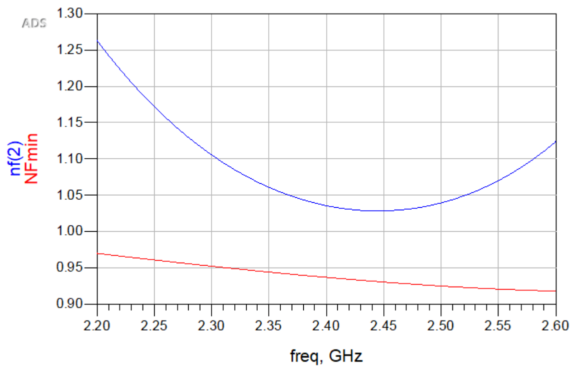

Fig. 8 Noise Figure Plot

F. Gain of the Low Noise Amplifier

The gain of LNA is greater than 15 dB over the frequency range 2.2 to 2.6 GHz

Figure 9 is the simulation result of our design.

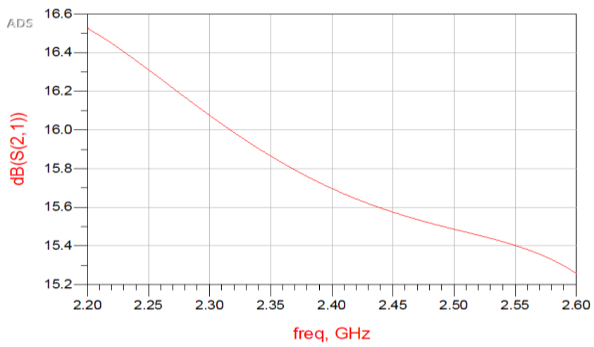

Fig. 9 Gain Plot

IV. RESULT IN TABULAR FORM

The following is the table of values of parameters of LNA at the different frequency points over the frequency range of 2.2 to 2.6 GHz. The circuit is unconditionally stable over the Bandwidth of 400

Frequency	*NF*	*NF_{min}*	*S_{21}*	*S_{11}*	*S_{22}*
2.20 GHz	1.263	0.970	16.526	-13.274	-18.897
2.25 GHz	1.172	0.961	16.311	-14.095	-17.211
2.30 GHz	1.105	0.952	16.075	-13.833	-16.828
2.35 GHz	1.061	0.944	15.864	-13.527	-17.097
2.40 GHz	1.035	0.937	15.697	-13.583	-17.608
2.45 GHz	1.028	0.930	15.574	-14.712	-18.266
2.50 GHz	1.039	0.925	15.456	-15.486	-19.414
2.55 GHz	1.070	0.921	15.402	-17.426	-21.915
2.60 GHz	1.124	0.918	15.260	-17.198	-25.147

Here,

NF: Noise figure in dB

S21: Gain in dB

S11: Input return loss in dB

S22: Output return loss in dB

V. CONCLUSION

The responses were satisfactory; Although we have scopes to improve the performance parameters. Within the bandwidth of 400 MHz, we got decent values of return losses. For the noise calculation, the temperature is considered at 16.85 degrees instead of 25 degrees Celsius as per IEEE standards because the manufacturing of these circuits takes place in the western world. To achieve stability easiest way is to place resistance at these 2 ports input and output. But we came to know that the value of resistance should be restricted as a higher value of resistance will suck most of the power. So when we are dealing with LNA we should never include any lossy component at the input of our LNA. It is good to include resistance at the output, because of the fundamental of Frii's formula. The approach of preferring unconditional stability over conditional stability will have certain advantages. If we can stabilize the device in the entire frequency band then we are free to do whatever we want and we are not worried about the device going into or circuit going into the stability region. But whenever we deal with conditional stability that means we are only stabilizing within the frequency range. For conditional stability, It is recommended to plot load stability circles and source stability circles to make sure that we don't venture into that unstable region. In this design, we use distributive components instead of Lumped components as it has the advantage of wider bandwidth for matching. That's how we got 400 MHz bandwidth. In the future, various bandpass filters can integrate with LNA for a more selective and more improvised performance in Gain and NF. The LNA implemented here is very useful in Radio Frequency signal-processing applications, in the front-end transceiver, WLAN, and ISM.

Retrieval Number: 100.1/ijitee.C97600211322
DOI: 10.35940/ijitee.C9760.0111322
Journal Website: www.ijitee.org

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)

The knowledge gained from this work will help to design the whole RF receiver system in the ISM band.

REFERENCES

1. B. Razavi, "Cognitive Radio Design Challenges and Techniques," in IEEE Journal of Solid-State Circuits, vol. 45, no. 8, pp. 1542-1553, Aug. 2010, doi: 10.1109/JSSC.2010.2049790.
2. Rafik M. Et-Trabelsi "Further Gained Power from GaAs E-pHEMT Low Noise Amplifier". MSc Dissertation, Electrical, and Computer Engineering Department, School of Engineering and Applied Sciences, Libyan Academy of Graduate Studies, Tripoli, Libya, July 2009.
3. Z. Pan, C. Qin, Z. Ye and Y. Wang, "A Low Power Inductorless Wideband LNA With Gm Enhancement and Noise Cancellation," in IEEE Microwave and Wireless Components Letters, vol. 27, no. 1, pp. 58-60, Jan. 2017, doi: 10.1109/LMWC.2016.2629969.
4. Joshi, Aniket & Mahajan, Shrinivas & Joshi, Bhal. (2012). Design and development of Low Noise Amplifier for RF/MW Receiver. International Journal of Scientific Research. 2. 224-227. 10.15373/22778179/JUNE2013/71.
5. Guillermo Gonzalez "Microwave Transistor Amplifier". 2nd ed. Prentice-Hall 1997.
6. A. Nadia, H. Belgacem and F. Aymen, "A low power low noise CMOS amplifier for Bluetooth applications," 2013 International Conference on Applied Electronics, 2013, pp. 1-4.
7. N. Shrivastava and R. Khatri, "Design of a 2.4-GHz differential low noise amplifier using 180nm technology," 2017 International Conference on Recent Innovations in Signal processing and Embedded Systems (RISE), 2017, pp. 494-497, doi: 10.1109/RISE.2017.8378206.
8. Jeonghoon Lee and Youngsik Kim, "CMOS low noise amplifier design techniques using shunt resistive feedback," 2005 Asia-Pacific Microwave Conference Proceedings, 2005, pp. 4 pp.-, doi: 10.1109/APMC.2005.1606587.
9. Z. Chen, H. Gao, R. van Dommele, D. Milosevic and P. G. M. Baltus, "Poster: Design consideration of 60 GHz low power low-noise amplifier in 65 nm CMOS," 2016 Symposium on Communications and Vehicular Technologies (SCVT), 2016, pp. 1-4, doi: 10.1109/SCVT.2016.7797670.
10. G. T. Bharathy, S. Bhavanisankari, T. Tamilselvi and G. Bhargavi, "Analysis and Design of a Novel Microstrip Filter for C - Band Applications," 2019 International Conference on Smart Structures and Systems (ICSSS), 2019, pp. 1-4, doi: 10.1109/ICSSS.2019.8882868.
11. Pozar, D.M., 'Microwave Engineering',(John-Wiley,2008, Ed. 3), pp.486-576
12. C. Füzy and A. Zólomy, "Design of broadband complex impedance-matching networks and their applications for broadbanding microwave amplifiers," 18-th INTERNATIONAL CONFERENCE ON MICROWAVES, RADAR AND WIRELESS COMMUNICATIONS, 2010, pp. 1-4.
13. https://html.alldatasheet.com/html-pdf/64591/HP/ATF-21170/23/1/ATF-21170.html

AUTHORS PROFILE

Abhay Chopde, is a Faculty in E & TC Engineering, VIT Pune for the past 24 years. He did PG and Ph.D. with specialization in Microelectronics from IIT Bombay. His main research areas are Semiconductor Devices, Electronic Circuit Design, Solar PV. He worked on various research consultancy projects from Industry as well as academia. Have guided more than 100 plus projects for UG as well as PG students. Email: abhay.chopde@vit.edu

Prashik Sadar, is a student pursuing a BTech in Electronics and Communication from Vishwakarma Institute of Technology, Pune. his areas of interest are Semiconductor Devices and Analog Electronics. He did two projects in the domain of Digital systems and one project of Antenna Design. He is looking forward to pursuing career in the Microelectronics and VLSI domain. Email: prashiksadar@gmail.com

Ashutosh Sabale, is a student pursuing a BTech in Electronics and Communication from Vishwakarma Institute of Technology, Pune. His area of interest is CMOS RF design, Antenna Design, and Digital Signal Processing. He has done 4 projects in the domain of Digital Electronics. He has published 2 papers so far. He is also having fond of coding and did some projects in that area. Email: ashutoshsabale19@gmail.com

Piyush Thite, is a student pursuing a BTech in Electronics and Communication from Vishwakarma Institute of Technology, Pune. His area of interest is Industrial Automation, Internet of Things, Embedded systems, and CMOS RF Design. He has published one paper so far. He is currently doing an internship in Seagate Technology. Email: piyush.thite@gmail.com

Raghvendra Zarkar, is a student pursuing a BTech in Electronics and Communication from Vishwakarma Institute of Technology, Pune. His area of interest is Digital Electronics, semiconductors. And he likes to explore in the future scope of Electronics developing area. Email: zraghvendra121@gmail.com

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)