

Analysis and Comparative Study of Effect of Feedback on BJT and CMOS Amplifier using Tanner Tools

Arpan Shah, Ankit Kapil, Rekha Agarwal, Sandhya Sharma

Abstract: - Feedback is a method in which a portion of the output returned to the input in order to modify the characteristics of the device. Feedback can be applied to transistor amplifier circuits to modify their performance characteristics such as gain, bandwidth, input and output impedance etc.. There are number of ways by which a signal can be derived from output and can be returned to input.

1. In this paper I will analyze and compare the different characteristics of Feedback and without feedback amplifier circuit of BJT.
2. After that I will compare the feedback amplifier of BJT with feedback amplifier of CMOS.
3. Different waveforms will be finding out by manipulating different parameters in the circuit design and mathematical calculation and comparison will be done.

Keywords: Feedback, Gain, Slew Rate, Amplifier, Power consumption.

I. INTRODUCTION

An amplifier in which feedback is incorporated known as feedback amplifier. Feedback can divide in two categories depending upon the phase of the returned (feedback) signal with respect to the input signal. If the returned signal is in phase with input signal, feedback is known as positive feedback. It increases the gain of the amplifier but reduces the bandwidth and stability of the circuit. It is used to produce oscillation. If the feedback signal is out of phase with respect to the input signal, it is known as a negative feedback.

Negative feedback improves the performance of an amplifier but reduces the Overall gain. It helps to stabilize the gain, increases bandwidth: reduces distortions and assures the repeatability of the circuit performance.

II. EDA TOOL

2.1 Objectives of the Tool

1. To develop an understanding of design and simulation of analog and digital logic circuits.
2. To get a basic understanding of layout of electronic circuits.
3. We will use Tanner tools for design and simulation.
4. This presentation introduces us to Tanner tools.
5. Analog and Transient Waveforms can be easily understood.

Manuscript received on November, 2012.

Arpan Shah, Student, Suresh Gyan Vihar University, Jaipur, India.

Ankit Kapil, Student, Suresh Gyan Vihar University, Jaipur, India.

Rekha Agarwal, Asst. Prof., Suresh Gyan Vihar University, Jaipur, India.

Sandhya Sharma, Asso. Prof., Suresh Gyan Vihar University, Jaipur, India.

2.2 Introduction to Tanner Tools.

1. Design is carried out with the help of SPICE.
2. SPICE-Simulation Program with Integrated- Circuit Emphasis.
3. It is basically used to analyze microelectronic circuits.
4. With the help of this we can analyze no of parameters which makes it very useful.
5. It is manufactured by Microsim Corporation for commercial purpose.

2.3 Types of Analysis That Can Be Performed

Three basic types of analysis are:

1. DC ANALYSIS
2. TRANSIENT ANALYSIS
3. AC ANALYSIS

Some other types of analysis are:

4. SENSITIVITY ANALYSIS
5. TEMPERATURE ANALYSIS
6. NOISE ANALYSIS
7. FOURIER ANALYSIS

III. CIRCUIT DESCRIPTION AND RESULTS:

3.1 BJT Amplifier without Feedback

I am taking the Common Emitter amplifier as a BJT amplifier without feedback. The circuit of common emitter amplifier (CE) shown below. I am analysis the small signal parameter of this amplifier, such as input resistance R_i , output resistance R_o , Voltage gain A_v and current gain A_i , using AC analysis facility of spice. These results will then compare with feedback amplifier. The transistor will be assumed to be modeled after the commercial 2N2222A NPN transistor.

To begin, let us compute the small-signal parameters of the CE amplifier. To obtain all four parameters, R_i , R_o , A_v and A_i , we will have to run two separate spice analysis one for computing the input current and the output Voltage for a known voltage applied to the input of the amplifier and the other for computing the current supplied by a voltage source connected to the output terminal of the amplifier when the input voltage source is set to zero. To monitor the current flowing through the load resistor, we placed a zero-valued voltage source in series with it, the input current of the amplifier can be determined by monitoring the current supplied by the input voltage source v_s .

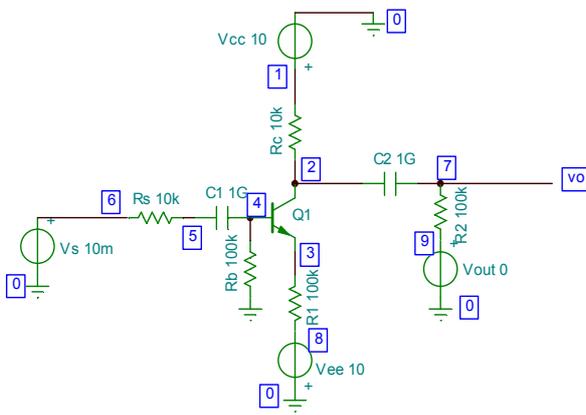


Figure 3.1 Common Emitter Amplifier (BJT without Feedback)

The four parameters of the amplifier can then be computed from these results according to the following formulas:-

$$Av = \frac{Vo}{Vs}$$

$$Ai = \frac{Io}{Ii}$$

$$Ri = \frac{Vi}{Ii}$$

$$Ro = \frac{Vo}{Io}$$

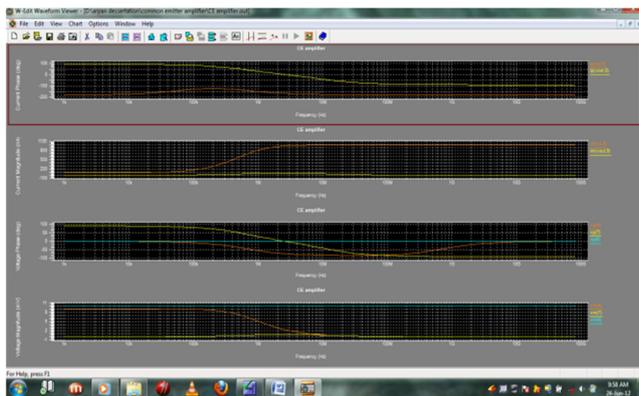


Figure 3.2 Common Emitter amplifier voltage and current waveform

3.1.1 Common Emitter amplifier AC Analysis (1KHz frequency)

Analysis of voltage gain

$$Vm(6) = 1.000E-02 \quad Vp(6) = 0.000E+00$$

$$Vm(7) = 5.068E-01 \quad Vp(7) = 1.799E+02$$

So we find that this amplifier has a midband voltage gain

$$Av = \frac{Vm(7)}{Vm(6)} = 50.68$$

Analysis of current gain

$$Im(Vs) = 6.803E-07 \quad Ip(Vs) = -1.796E+02$$

$$Im(Vout) = 5.068E-05 \quad Ip(Vout) = 1.799E+02$$

So we find that this amplifier has a midband current gain

$$Ai = \frac{Im(Vout)}{Im(Vs)} = -74.49$$

Analysis of input resistance

$$Vm(4) = 3.198E-03 \quad Vp(4) = -9.300E-01$$

$$Im(Vs) = 6.803E-07 \quad Ip(Vs) = -1.796E+02$$

So we find that this amplifier has a midband input resistance

$$Ri = \frac{Vm(4)}{Im(Vs)} = 4.7K\Omega$$

To determine the output resistance of this amplifier we repeat this same process but set the level of the input voltage source to 0V and increase the level of the voltage source in series with the load resistance to 10mV AC.

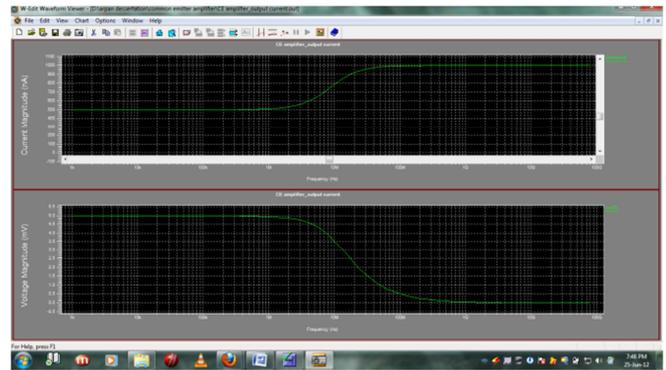


Figure 3.3 Output Voltage and Output Current waveform

Analysis of output resistance

$$Vm(7) = 4.728E-03 \quad Vp(7) = 0.000E+00$$

$$Im(Vout) = 5.272E-07 \quad Ip(Vout) = 1.800E+02$$

So we find that this amplifier has a midband input resistance

$$Ro = \frac{Vm(7)}{Im(Vout)} = 8.97 K\Omega$$

3.1.2 Common Emitter amplifier Slew Rate analysis

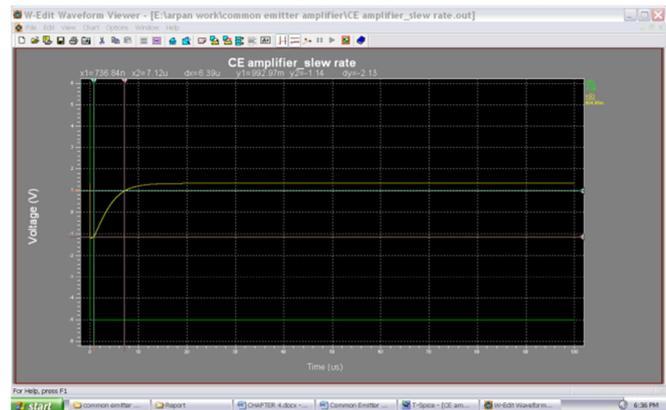


Figure 3.4 Slew rate of common emitter amplifier.

Slew rate of BJT common emitter amplifier can be calculated from the formula dv/dt

$$V2 - V1 / T2 - T1 = 2.13 \text{ v} / 6.39 * 10^{-6} = .33 \text{ V/microsecond}$$

4.1.3 Common Emitter amplifier power consumption analysis

The total power dissipation can be calculated by $P=V*I$. Here current at Power supply is from +10 volts to -10 volts that is equal to 20 volts and current at Vcc is 5.000E-04 so power can be calculated as:

$$\text{Power dissipation} = 20 * .5 * 10^{-3} = 10mW$$

3.2 BJT Feedback Amplifier

The BJT feedback amplifier circuit has shown below. I am analysis the small signal parameter of this amplifier, such as input resistance R_i , output resistance R_o , Voltage gain A_v and current gain A_i , using AC analysis facility of spice. These results will then compare with without feedback amplifier. The transistor will be assumed to be modeled as a commercial BJT transistor.

To begin, let us compute the small-signal parameters of the BJT feedback amplifier. To obtain all four parameters, R_i , R_o , A_v and A_i , we will have to run two separate spice analysis one for computing the input current and the output Voltage for a known voltage applied to the input of the amplifier and the other for computing the current supplied by a voltage source connected to the output terminal of the amplifier when the input voltage source is set to zero. To monitor the current flowing through the load resistor, we placed a zero-valued voltage source in series with it. The input current of the amplifier can be determined by monitoring the current supplied by the input voltage source V_s .

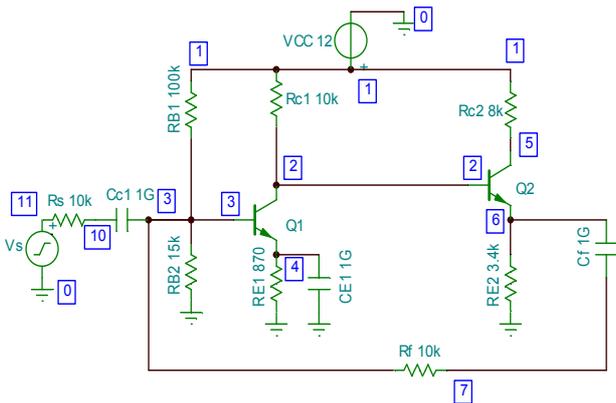


Figure 3.5 BJT Feedback amplifier Circuit

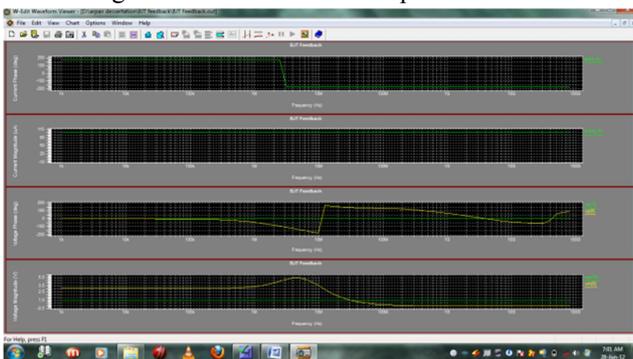


Figure 3.6 BJT feedback amplifier Input, Output Voltage and Input current waveform

3.2.1 BJT Feedback amplifier AC Analysis (1KHz frequency)

Analysis of voltage gain

$$V_m(11) = 1.000E+00 \quad V_p(11) = 0.000E+00$$

$$V_m(5) = 3.0815E+00 \quad V_p(5) = -47.383E+06$$

So we find that this amplifier has a voltage gain

$$A_v = \frac{V_m(5)}{V_m(11)} = 3.08$$

Analysis of input resistance

$$V_m(11) = 1.000E+00 \quad V_p(11) = 0.000E+00$$

$$I_m(vin) = 95.676E-06 \quad I_p(vin) = 180.000E+00$$

So we find that this amplifier has a voltage gain

$$R_i = \frac{V_m(11)}{I_m(vin)} = 10.4519 \text{ K}\Omega$$

Analysis of current gain

$$I_m(vin) = 95.676E-06 \quad I_p(vin) = 180.000E+00$$

$$I_m(vo) = 9.7167E-03 \quad I_p(vo) = -180.000E+00$$

So we find that this amplifier has a voltage gain

$$A_i = \frac{I_m(vo)}{I_m(vin)} = 101.56$$

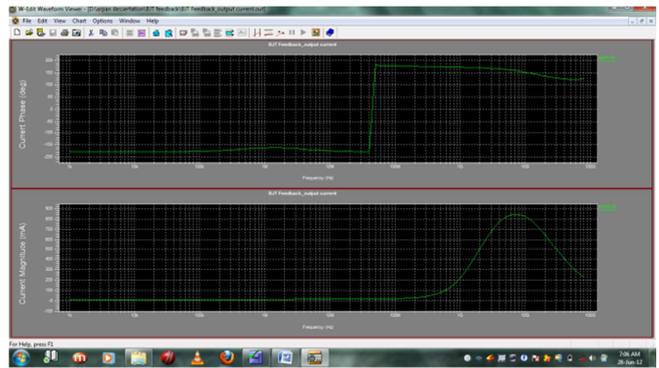


Figure 3.7 BJT feedback amplifier Output current waveform

Analysis of Output resistance

$$V_m(5) = 3.0815E+00, \quad V_p(5) = -47.383E+06$$

$$I_m(vo) = 9.7167E-03 \quad I_p(vo) = -180.000E+0$$

So we find that this amplifier has a midband voltage gain

$$R_o = \frac{V_m(5)}{I_m(vo)} = 317.13\Omega$$

3.2.2 BJT Feedback Amplifier Slew Rate analysis

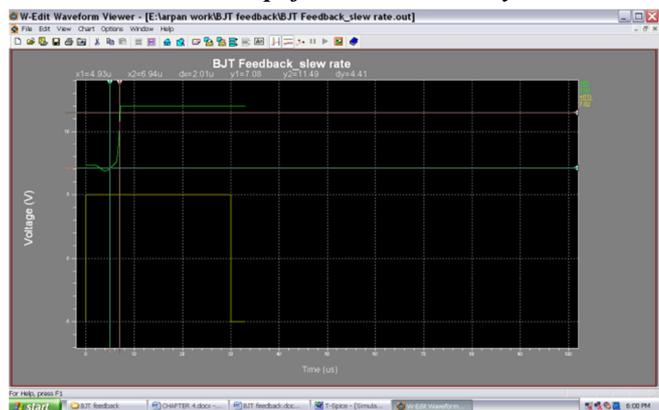


Figure 3.8 Slew rate of BJT feedback amplifier

Slew rate of BJT feedback amplifier can be calculated from the formula dv/dt

$$V_2 - V_1 / T_2 - T_1 = 4.41 \text{ v} / 2.01 * 10^{-6} = 2.194$$

V/microsecond

3.2.3 BJT Feedback Amplifier power consumption analysis

The total power dissipation can be calculated by $P=V*I$. Here current at Power supply is from +12 volts to 0 volts that is equal to 12 volts and current at V_{cc} is $1.626E-03$ so power can be calculated as:

$$\text{Power dissipation} = 12 * 1.626 * 10^{-3} = 19.512\text{mW}$$

3.3 CMOS Feedback Amplifier

The CMOS feedback amplifier circuit shown below. I am analysis the small signal parameter of this amplifier, such as input resistance R_i , output resistance R_o , Voltage gain A_v and current gain A_i , using AC analysis facility of spice. These results will then compare with without feedback amplifier. The transistor will be assumed to be modeled as a commercial CMOS transistor with $\lambda=0$.

To begin, let us compute the small-signal parameters of the CMOS feedback amplifier. To obtain all four parameters, R_i , R_o , A_v and A_i , we will have to run two separate spice analysis one for computing the input current and the output Voltage for a known voltage applied to the input of the amplifier and the other for computing the

current supplied by a voltage source connected to the output terminal of the amplifier when the input voltage source is set to zero. To monitor the current flowing through the load resistor, we placed a zero-valued voltage source in series with it, the input current of the amplifier can be determined by monitoring the current supplied by the input voltage source V_s .

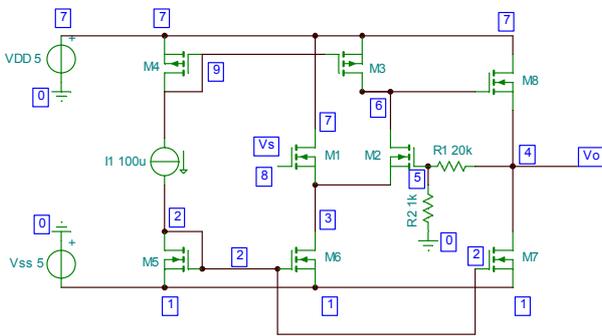


Figure 3.9: CMOS Feedback amplifier Circuit

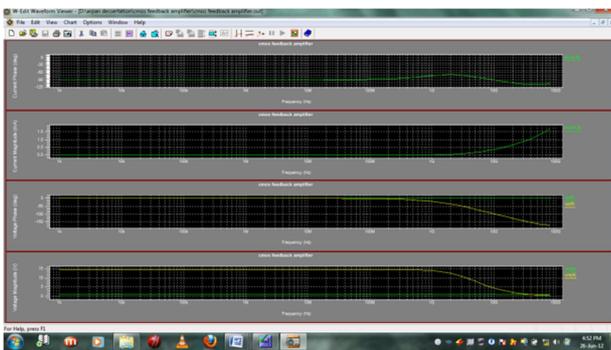


Figure 3.10 CMOS Feedback amplifier input, output Voltage & input Current

3.3.1 CMOS Feedback amplifier AC Analysis (1KHz frequency)

Calculation for voltage gain

$$V_m(8) = 1.000E+00 \quad V_p(8) = 0.000E+00$$

$$V_m(4) = 14.5790E+00 \quad V_p(4) = -19.292E-06$$

So we find that this amplifier has a midband voltage gain

$$A_v = \frac{V_m(4)}{V_m(8)} = 14.58$$

Calculation for input resistance

$$V_m(8) = 1.000E+00 \quad V_p(8) = 0.000E+00$$

$$I_m(vin) = 16.400E-12 \quad I_p(vin) = -90.000E+00$$

So we find that this amplifier has a midband voltage gain

$$R_i = \frac{V_m(8)}{I_m(Vin)} = 60.976G\Omega$$

Calculation for current gain

$$I_m(vin) = 16.400E-12 \quad I_p(vin) = -90.000E+00$$

$$I_m(vo) = 19.8710E-03 \quad I_p(vo) = 180.000E+00$$

So we find that this amplifier has a midband voltage gain

$$A_i = \frac{I_m(Vo)}{I_m(Vin)} = 1.211x10^9$$

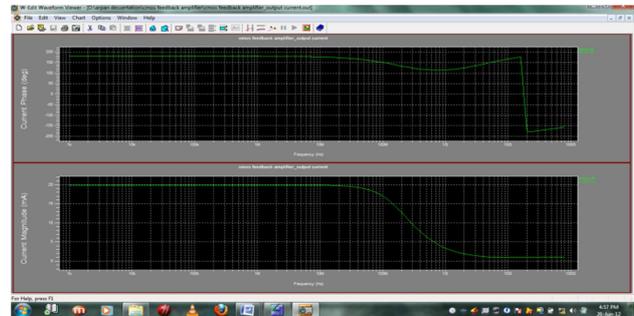


Figure 3.11 CMOS Feedback amplifier output Current

Calculation for Output resistance

$$V_m(4) = 14.5790E+00 \quad V_p(4) = -19.292E-06$$

$$I_m(vo) = 19.8710E-03 \quad I_p(vo) = 180.000E+00$$

So we find that this amplifier has a midband voltage gain

$$R_o = \frac{V_m(4)}{I_m(Vo)} = 733.68\Omega$$

3.3.2 CMOS feedback amplifier slew rate analysis

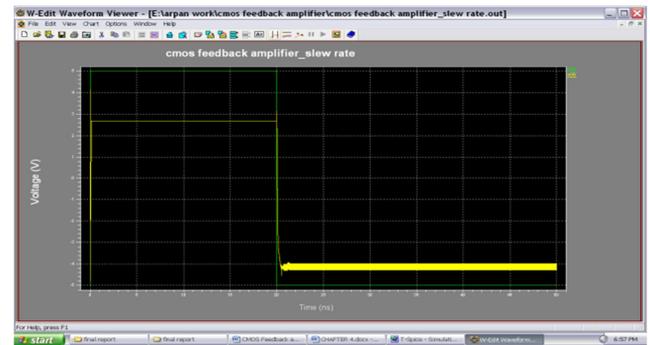


Figure 3.12 Slew rate of CMOS feedback amplifier

Slew rate of cmos feedback amplifier can be calculated from the formula dv/dt

$$V_2 - V_1 / T_2 - T_1 = 3.66 \text{ v} / 80.02 * 10^{-12} = .0457 \text{ V/ pico-second.}$$

3.3.3 CMOS feedback amplifier power consumption analysis.

The total power dissipation can be calculated by $P=V*I$. Here current at Power supply is from +5 volts to -5 volts that is equal to 10 volts and current at V_{cc} is $5.9408E-04$ so power can be calculated as:

$$\text{Power dissipation} = 10 * .59408 * 10^{-3} = 5.9408 \text{ mW}$$

IV. RESULTS

Mathematical representation of Comparative study

The mathematical comparison between different amplifier circuits can be described as follow:

Table 1 Comparative table of different parameter of Common Emitter amplifier BJT feedback amplifier and CMOS feedback amplifier.

S.No.	Parameters	Common Emitter amplifier	BJT feedback amplifier	CMOS feedback amplifier
1	Voltage Gain	50.68	3.08	14.58
2	Current Gain	-74.49	101.56	$1.211x10^9$
3	Input Resistance	4.7k Ω	10.4519 k Ω	60.976 G Ω
4	Output Resistance	8.97k Ω	317.13 Ω	733.68 Ω
5	Slew Rate	.33 V/micro second	2.194 V/micro second	.0457 V/Pico second
6	Power dissipation	10 mW	19.512 mW	5.9408 mW

V. CONCLUSION

Now a days electronics circuit has major role of feedback. The project I have undertaken can be used as a reference or as a base for realizing the feedback circuit so it can be further implemented in other projects of greater level. In this report I worked in two parts first I compared the BJT amplifier circuit with the BJT feedback amplifier where I found that feedback is good to achieve less voltage gain and high current gain on the cost of high slew rate and more power consumption. In second part of my work I compared the BJT feedback amplifier with the CMOS feedback amplifier where I observed that CMOS feedback circuit is faster than BJT feedback circuit with a low power consumption. Therefore due to the importance of both the devices we switched on the new technology which is very efficient and well known now days that is Bi-Cmos technology. So we can further work on lots of researches in this field in which we can try to improve all the parameters further by utilizing the advantage of both the previous research results and can study further to enhance the capacity and efficiency of feedback circuit.

REFERENCE

- [1] K. Wang, P. C. Huang and C. Y. Huang, "A Fully Differential CMOS Transconductance-Transimpedance Wideband Amplifier," IEEE Trans. Circ. and Syst. II, vol. 42, no. 11, pp. 745-748, November 1995.
- [2] TSENG S.-H., HUNG Y.-J., JUANG Y.-Z., LU M. S.-C., A 5.8-GHz VCO with CMOS-compatible MEMS inductors, Sensors and Actuators A: Physical, Vol. 139, pp. 187-193, 2007.
- [3] Sam Ben-Yaakov "A Unified Approach to Teaching Feedback in Electronics Circuits Courses" IEEE Transaction on education VOL. 34 no. 4, NOV. 1991
- [4] NAGEL, L.W.: 'SPICE 2: A computer program to simulate semiconductor circuits', ERL Memo ERL-M520, Electronics Research Laboratory, University of California, Berkeley, May 1975
- [5] VLADIMIRESCU, A., NEWTON, A.R., and PEDERSON, D.O.: 'SPICE version 2G.0, user's guide', University of California, Berkeley, Sept. 1980.
- [6] Gianna De Rubertis, Dept. of Chem. Eng. & Appl. Chem., Univ. of Toronto, Ontario, Canada
Davies, S.W. NanoBioscience, IEEE Transactions on Dec. 2003
- [7] A.P. Malvino, Electronic Principles (2nd Ed. 1979. ISBN 0-07-039867-4)
- [8] Jacob Millman, Microelectronics: Digital and Analog Circuits and Systems, McGraw-Hill, 1979, ISBN 0-07-042327-X,
- [9] Horowitz, Paul; Hill, Winfield (1989). The Art of Electronics. Cambridge, UK: Cambridge University Press.
<http://books.google.com/books?id=bkOMDgwFA28C&pg=PA177&lpg=PA177#v=onepage&q&f=false>.

ARPAN SHAH, Student of M.Tech Final year in Suresh Gyan Vihar University, Jaipur. I have completed my B.E degree in 2008 from Rajasthan University. I am currently working in the VLSI field.

ANKIT KAPIL, Student of M.Tech Final year in Suresh Gyan Vihar University, Jaipur. I have completed my B.E degree in 2009 from Rajasthan University. I am currently working in the VLSI field.

REKHA AGARWAL, M.TECH degree in VLSI from Banasthali Vidhyapith, Jaipur. She is currently working as Asst. Prof. in Dept. of Electronics and Communication Engg. In Suresh Gyan Vihar University, Jaipur.

SANDHAYA SHARMA, M.TECH degree in Digital and wireless Communication from MBM, Jodhpur. She is currently working as Asso. Prof. in Dept. of Electronics and Communication Engg. In Suresh Gyan Vihar University, Jaipur.