Design of OPAMP Based Applications using 120 nm Technology

Gohil Dharmendra R.[#1], Mohd. Abuzer Khan[*2]

[#]*Electronics & Communication, R.G.P.V Bhopal*

L.K.C.T. Indore M.P. India

[1]erdrgohil@gmail.com

[2]abuzerlkct@gmail.com

Abstract-This paper introduces a generalized method for simulating the various applications of operational amplifier generally termed as op-amps using TSpice. Now days engineering education is taught using simulation software. The usefulness of this SPICE software in learning various application of op-amp is enormous than proto typing practical laboratory. This PSPICE software's which are important part of engineering education plays significant role in analyzing and simulating the various applications of op-amp such as summing, integrator and comparator. We will present some designing, analysis and simulation of specific applications of op-amp using this software's in detail which plays a vital role in various field of engineering.

Keywords— Nmos, PmosOperational amplifier, TSpice, SPICE, software.

Introduction

Operational amplifier in present-day is the most widely used electronics device in field of electronic systems such as communications systems, medical electronics, instrumentation, signal processing and performing many mathematical operations such as addition, subtraction, multiplication, integration and differentiations etc. It consists of various stages namely input stage, intermediate stage gain, level shifting stage and output stage and is fabricated as an integrated circuit. The symbol of typical op-amp is shown in fig.1.
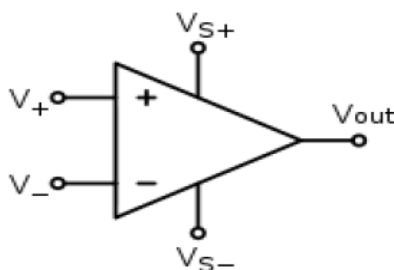

Fig1. Circuit symbol of op-amps

It has two input terminal named inverting V- and non inverting terminal V+ and one output terminal Vout. Vs+ and Vs- are power supply terminal. The ideal operational amplifier characteristics are:

1. Infinite input impedance

2. Zero output impedance

3. Voltage gain is infinite

4. Bandwidth is also infinite

5. Slew rate is infinite

6. Common mode rejection ratio is infinite

7. input Offset voltage is zero

The IC 741 is widely used general purpose op-amp whose circuit symbol is shown in fig 1.The Op-amp can be used in two configurations namely Open loop configuration and closed loop configuration. The open loop configuration is one in which no feedback in any form is fed to the input from the output whereas closed loop configuration is in which a fraction of output is fed back to input. A few limitations of open loop configurations are: 1. clipping of output waveform may occur if output voltage exceeds saturation voltage of op-amp. 2. Open loop gain is not constant 3. Bandwidth is negligibly small. That why this Configuration is not suitable for AC applications but it finds use in non linear applications such comparators, square wave generator and astable multivibrator. The closed loop configurations finds its utilization in linear applications such adder, transconductance amplifier, instrumentation amplifier, integrator, differentiator, log and antilog amplifier.

II. VARIOUS APPLICATIONS OF OP-AMP

1. Designing of Comparator

The comparator is one of the most critical parts of almost all analog-to-digital (ADC) converters, Depending up on their size and structure they can have a severe impact on the performance of a device. The speed and resolution of an ADC is directly affected by the input offset voltage, the

delay and input signal range (5).Comparators are classifieddepending upon there nature, functionality and inputs. Like voltage and current comparators, continuous and discrete time, etc.

Some of the basic application of comparators is analog-to-digital conversion, function generator, signal detection and neural networks etc.
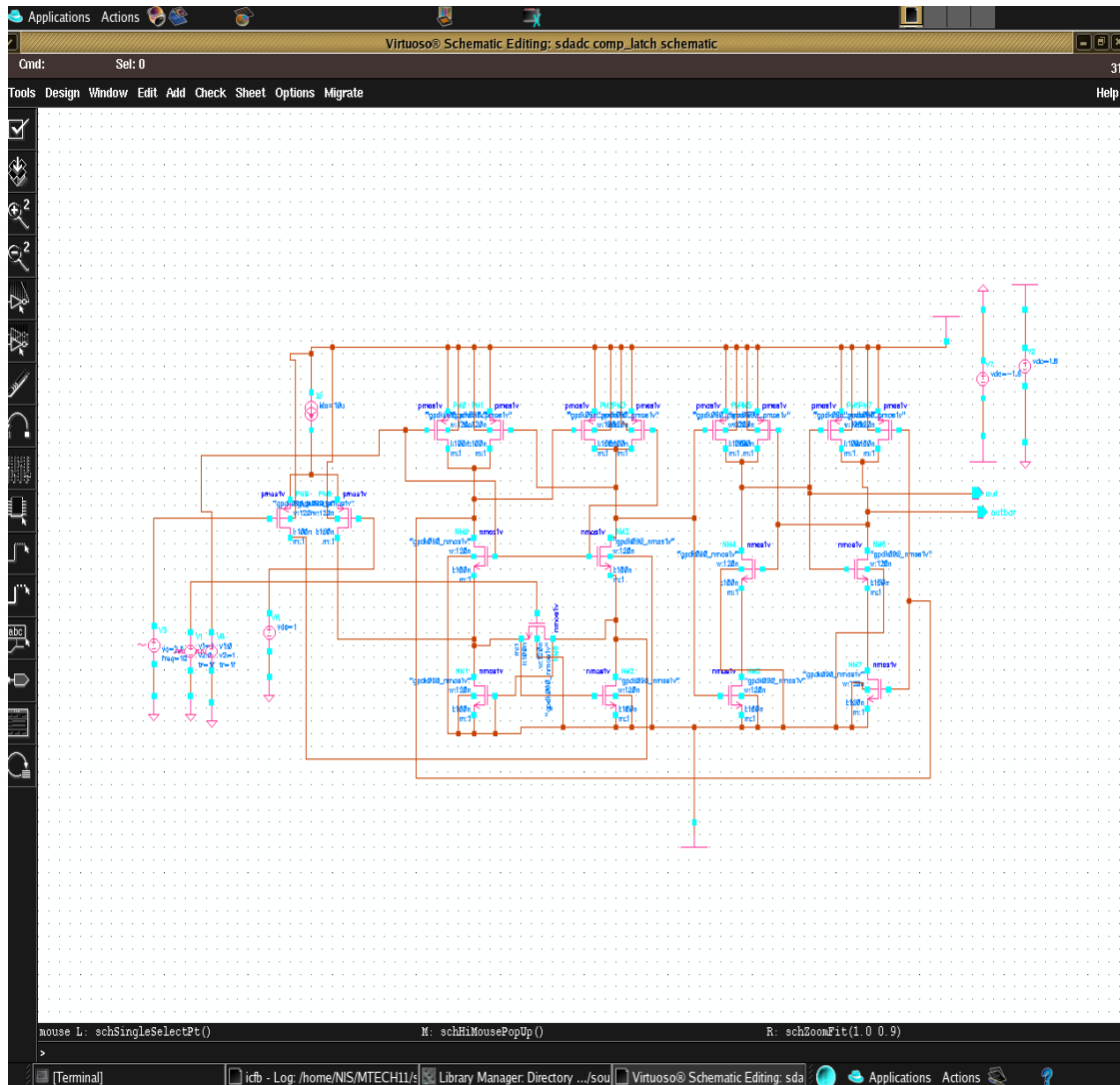

Figure 2: Schematic Design of latch-Comparator

The latch- comparator is simulated using a sinusoidal input signal to the differential pair in its one terminal and making the other ground, two non-overlapping and complementary clock signals are applied, which helps the comparator to improves its speed, as well as the sampling frequency of the quantizer, 3 shows the transient analysis of the latched comparator. Here the analog input signal is compared against last sample signal to see if it is higher than the reference or not. If it is larger than the output is increased else decreased.

The density of '1s' and '0s' forming a pulse stream at the output is the digital representation of the input analog signal. A converter needs a sampling frequency more than twice the signal frequency to reproduce the signal without distortion. The density of the pulses represents the average value of the input; most of the pulses are high for the positive peak of input signal and the density of negative pulses represents the negative peak of the signal.

International Journal of Research

Available at https://edupediapublications.org/journals

p-ISSN: 2348-6848
e-ISSN: 2348-795X
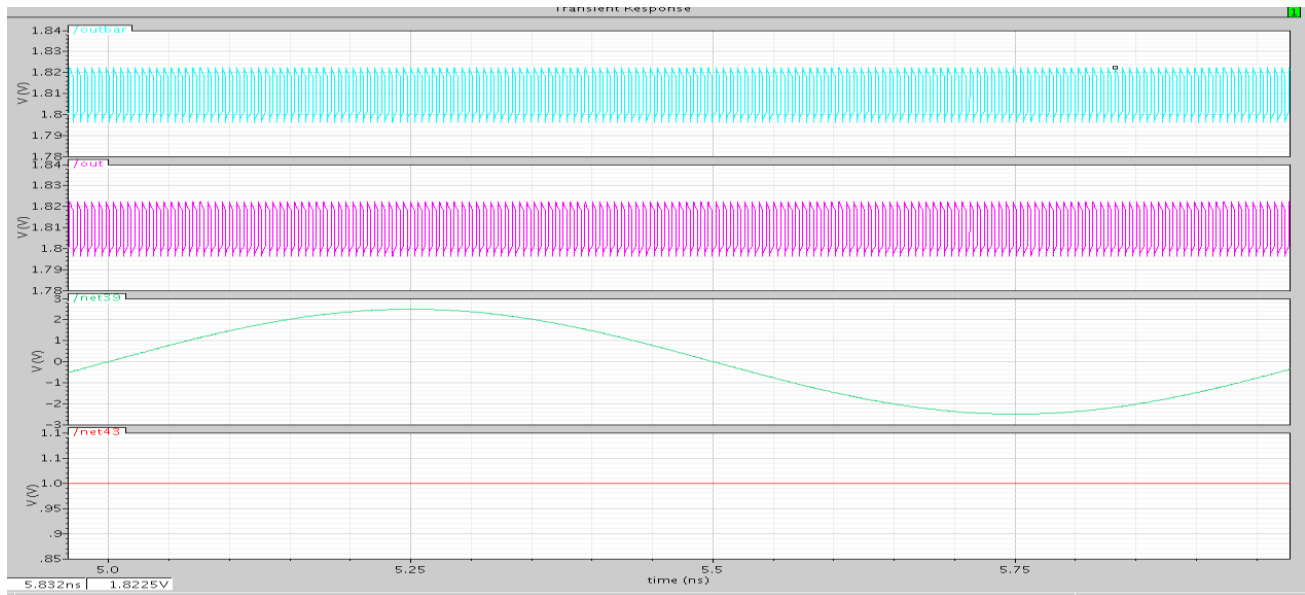**Volume 03 Issue 17**
November 2016

Figure 3: Simulated waveform of Comparator

2. Designing of Summing Amplifier

The summing amplifier is a very flexible circuit based on the standard operational amplifier, which can be used for combining multiple inputs. It has a single input voltage applied to the inverting input terminal. If more resistors are added to the input equal to the input resistor, we end up with another operational amplifier called as summing amplifier. Figure 4 shows the circuit of a summing amplifier
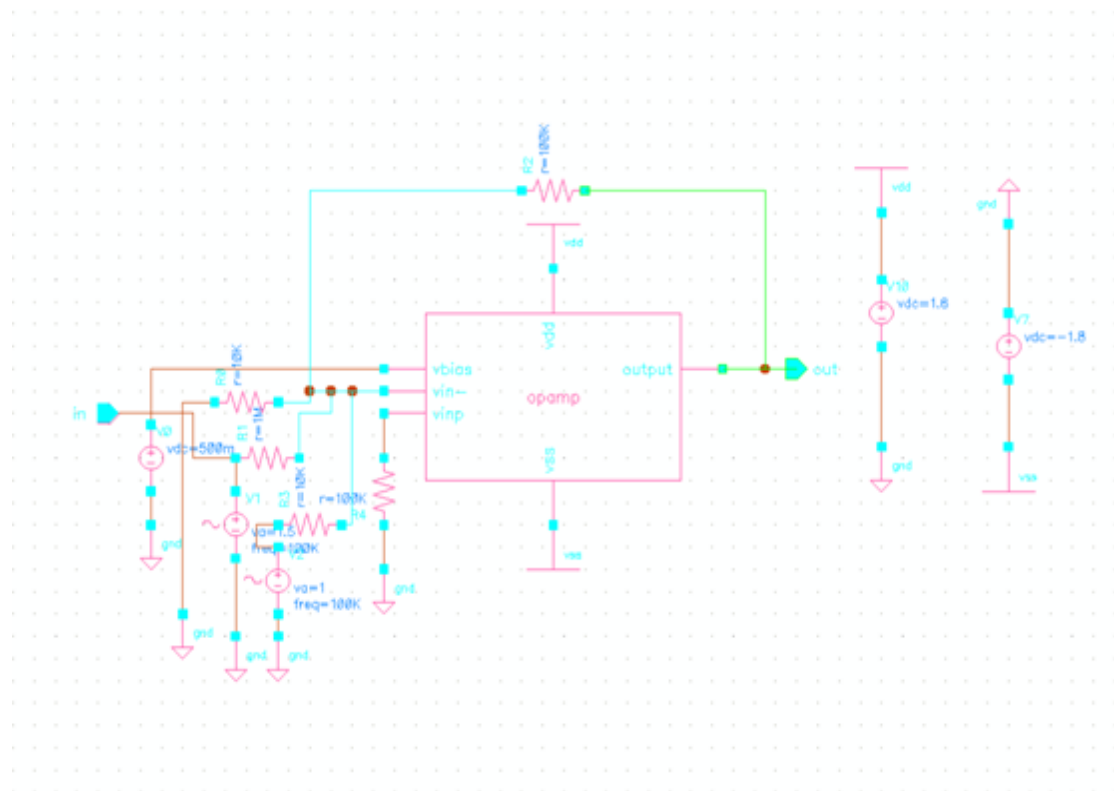

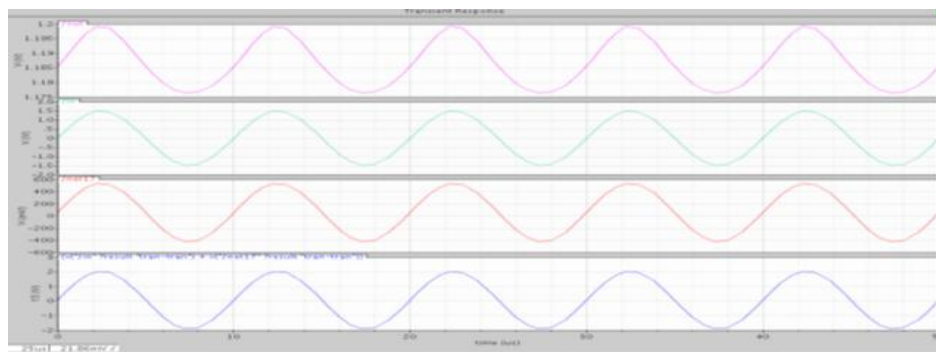Figure 4: Schematic of summing amplifier

Figuure 5: Simulation result of summing amplifier

3. Designing of Integrator

The integrator is designed using a two stage Op-Amp, in cadence 0.12um CMOS technology and is connected with a feed- back capacitor which helps in charging of the input voltage to give a integrated output. It is given a sinusoidal input signal from the summing amplifier whose output is the sum of the input and the feed-back signal and gives ramp output corresponding to a given sinusoidal input. The below 6 shows the structure of an integrator circuit and 7 gives the simulation result of the integrator.
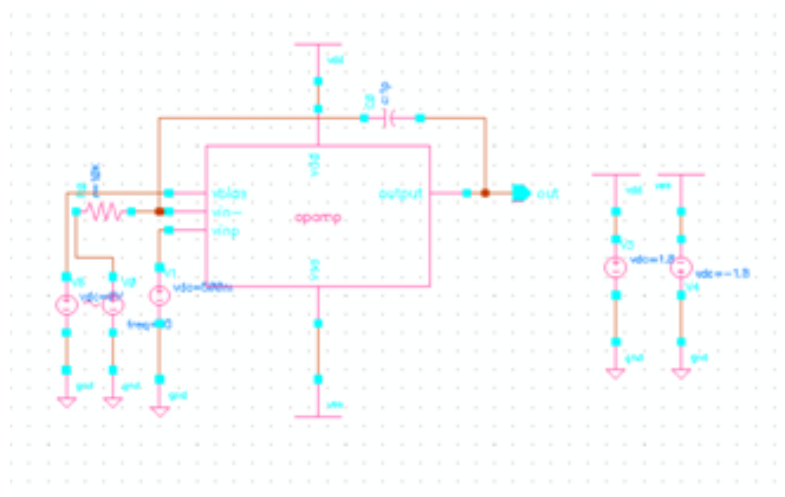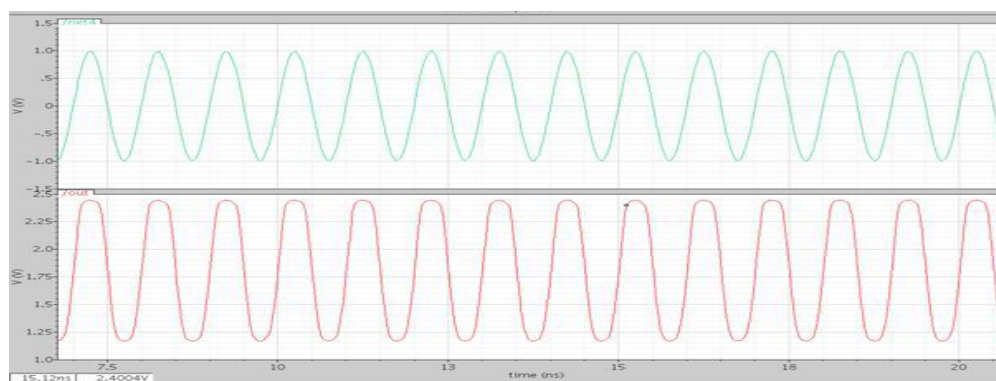

Figure 6 schematic of Integrator

Figure 7: Simulation Result of integrator

III.   CONCLUSION

This paper mainly discusses the various applications of 741 operation amplifier and their simulations using TSpice software. We can design, simulate and verify any of applications with required parameters. Because of graphical facilities of TSpice, it is useful in enhancing the understanding of various applications of op-amps. We can also use different spice software like LT spice, Top Spice and Spice Opus for better understanding of various applications and compare the results with theoretically calculated result.

REFERENCES:

[1] Boylestad R., L. Nashelsky, 1992. Electronics Devices and Circuit Theory, Prentice-Hall Inc.
[2] PSPICE A/D online manual

[3] National Semiconductor Company, National Operational Amplifier databook, 1995 [4] Rashid M.H. Microelectronics Circuit Analysis and Design, PWS Publishing Co, 1999
[5] Rashid M.H., Introduction to PSpice Using OrCAD for Circuits and electronics, Pearson Education., 2006

[6] Gayakwad, R.A, Op-amps and Linear integrated circuits, Prentice-Hall India, 2000.